UCRL-ID-131898

Working Group Meeting on Heavy Vehicle Aerodynamic Drag: Presentations and Summary of Comments and Conclusions

Rose McCallen, Walt Rutledge
Don McBride, Kambiz Salari
Walter Gutierrez, Fred Browand
Anthony Leonard, Jim Ross
Karlin Roth

September 28, 1998

Lawrence Livermore National Laboratory

Work performed under the auspices of the U.S. Department of Energy by the Lawrence Livermore National Laboratory under Contract W-7405-ENG-48.

Working Group Meeting on Heavy Vehicle Aerodynamic Drag:

Presentations and Summary of Comments and Conclusions

Jointly written by

Lawrence Livermore National Laboratory
Sandia National Laboratories
University of Southern California
California Institute of Technology
NASA Ames Research Center

Introduction

The first Working Group Meeting on Heavy Vehicle Aerodynamic Drag was held at Sandia National Laboratories (SNL) in Albuquerque, New Mexico on August 28, 1998. The purpose of the meeting was to review the proposed Multi-Year Program Plan (MYPP) and provide an update on the Group's progress. In addition, the technical details of each organization's activities were presented and discussed.

Presentations were given by representatives from the Department of Energy (DOE) Office of Transportation Technology Office of Heavy Vehicle Technology (OHVT), Lawrence Livermore National Laboratory (LLNL), SNL, University of Southern California (USC), California Institute of Technology (Caltech), and NASA Ames Research Center. These presenters are part of a DOE appointed Technical Team assigned to developing the MYPP.

The goal of the MYPP is to develop and demonstrate the ability to simulate and analyze aerodynamic flow around heavy truck vehicles using existing and advanced computational tools (A Multi-Year Program Plan for the Aerodynamic Design of Heavy Vehicles, R. McCallen, D. McBride, W. Rutledge, F. Browand, A. Leonard, J. Ross, UCRL-PROP-127753 Dr. Rev 2, May 1998).

This report contains the technical presentations (viewgraphs) delivered at the Meeting, briefly summarizes the comments and conclusions from the Meeting participants, and outlines the future action items.

The MYPP and Presentations

As described in the viewgraph presentations, the project plan is divided into two related and overlapping efforts:

Advanced Computations and Experiments of Benchmark Geometries

Evaluation of Current and New Technologies

Each effort has near-term deliverables as well as longer-term goals. The computations and experiments effort will provide rapid results for simple benchmark geometries, and will then advance to more complex geometries. The evaluation of current and new technologies will continue to provide assessment for promising emerging technology.

Attached is a list of the presentations delivered at the Meeting (see meeting agenda) and the viewgraphs presented are enclosed herein.

Summary Comments and Conclusions

MYPP and Budget

Past drafts of the MYPP have included a third effort:

Demonstration of a Device Integration Process

It was hoped that the demonstration of a a device integration process for an existing trailer add-on device would be a near-term effort, with the promise for a long-term impact.This task was omitted from the current draft of the MYPP because of budget constraints. The DOE funding representative, Sid Diamond, has requested that this effort be added back into the MYPP as a task that may be added in the future, if funding permits.

It is anticipated that we will receive 80 to 85% of our requested budget for FY99 and FY00. Our budget estimates are $635K and $1,233K, respectively. This funding is for the computations and experiments and evaluation of new technologies efforts described above and not for the additional demonstration effort.

Project Overview

For near-term impact the first benchmark case will involve the Sandia integrated tractor-trailer model. Comparisons will be made of Reynolds-Average Navier-Stokes (RANS) and Large-Eddy Simulations, as well as detailed experimental verification. Along with the baseline case of the integrated tractor-trailer, height mismatches and gap distances between the tractor and trailer will be investigated.

There are advantages in using the Sandia Model as the first benchmark case. It is a simple geometry with some existing data and some modeling has already been done. Thus, making it more likely that we will achieve a near-term impact with the existing budget constraints. In addition, the final results are not proprietary and can be made available for comparison to commercial software (e.g., a results comparison at a workshop).

The projected funding needs outlined in the Aero Team's budget assumed the use of leveraged funds for FY99 and FY00. However, more funds will be needed if less than the budgeted dollars are provided. Possibilities for other funding sources were suggested and action items are outlined below for further investigation of these possible sources.

Experiments

SNL will provide the results of experiments performed at the Texas A&M wind tunnel for the integrated model at Reynolds number, Re, or 1,600,000 ($Re = UL/\nu$, where U and L are characteristic velocity and length scale, respectively, and ν is the kinematic viscosity). Time-averaged results are provided from these tests. SNL is providing use of the Sandia Model for the future experiments at NASA Ames.

NASA Ames will perform detailed measurements for a range of Re on the Sandia Model in their 7 ft by 10 ft wind tunnel, providing full three-dimensional velocity field and surface pressure results. These results are being provided free of charge. Their second series of tests will be run with a donated model from Navistar International for a Re sensitivity study. These tests will be performed in the NASA Ames 12 foot wind tunnel at a range of Re up to 5,000,000. The 12 ft tunnel test will be accomplished at one-third cost.

USC will perform experiments at two Re within the range of 200,000 to 400,000 using the Sandia Model, with and without trailer-tractor height mismatch and gaps. Tunnel instrumentation will be provided using leveraged funds.

Computations

SNL will perform the RANS calculations for high and low Re cases of the Sandia Model. The LES for low Re with some attempt at high Re will be performed by LLNL using a finite element method and by Caltech using a vortex method approach.

Future Meetings and Workshops

It was suggested that the location of the Working Group Meetings rotate among the Aero Team's facilities. The next Working Group Meeting will be held at NASA Ames during the scheduled Sandia Model testing, which should occur in the December 1998 to February 1999 time frame. LLNL will assist NASA in the meeting planning.

DOE sponsors requested that the next Aero Drag Workshop be held in the Fall of 1999. LLNL will be responsible for the Workshop, but the entire Aero Team and DOE sponsors will be directly involved in the Workshop planning and organization.

Action Items

The follow-on prioritized action items with the individuals responsible for the tasks are as follows:

1. Distribute viewgraphs and meeting results. (R. McCallen)

2. Develop a combined project plan with milestones clearly showing the contribution of each organization and how all the contributions come together. (R. McCallen)

3. Schedule site visits to Paccar, Mack, and Schneider. (R. McCallen)

4. Start planning work shop for Fall 1999. Investigate the possibility of connecting it with and existing conference (e.g., Truck Maintenance Council meeting in October 1999, see SAE web page). (R. McCallen)

5. Plan next working group meeting at NASA Ames around January 1999. (J. Ross and K. Roth)

6. Add back into MYPP the Section on demonstration of a device integration process and distribute the MYPP for feedback first from Aero Team and DOE sponsors and them from industry and others. (R. McCallen)

7. Investigate California State funding sources. (F. Tokarz and F. Browand)

8. Draft letter of appreciation to Navistar International for their exceptional participation in our effort. (R. McCallen)

9. Publish results at SAE conferences (e.g., Technical Meetings in February). (All Aero Team members)

10. Investigate rumors of new Volvo integrated tractor trailer. (R. Wares)

11. Provide Aero Team with GTRI's project plan for preliminary review. (S. Diamond)

12. Investigate the possibilities of 'collaborators' (i.e., industry, universities, and laboratories). (J. Routbort)

- Agenda -

Truck Aero Team Meeting

Sandia National Laboratories, Albuquerque, NM

August 28, 1998

Purpose of Meeting

Review of plans

Update on progress

Technical details of approach and results

Introduction

Introduction to Sandia National Laboratories (Walt Rutledge)

Project and Budget Update (Sid Diamond)

Overview of Project Plan and Budget (Rose McCallen)

Experimental Work and Progress

Existing Data from Texas A&M (Walt Gutierrez)

Wind Tunnel Tests at USC (Fred Browand)

Work on New Model Designs (Fred Browand)

NASA 7'x10' and 12' Wind Tunnel Tests (Karlin Roth)

Computational Work and Progress

RANS and LES Modeling Plans and Results at SNL (Kambiz Salari)

FEM and LES Development and Modeling Plans at LLNL (Rose McCallen)

Vortex Method and LES Development and Modeling Plans at Caltech (Tony Leonard)

Evaluation of New Technologies

Discussions

Wrap-up Discussion

Calendar of Near Term Events (e.g., Site Visits, Next Progress Meeting, Experiments)

Near Term Action Items

AERODYNAMICS DRAG MEETING
SANDIA NATIONAL LABORATORIES
FRIDAY, AUGUST 28, 1998

Attendance List

Attendee	Organization	Contact Addresses
Sid Diamond	DOE/OTT/OHVT	Tel: (202)586-8032 FAX: (202)586-1600 e-mail: sid.diamond@ee.doe.gov
Frank Tokarz	LLNL	Tel: (925)423-3459 FAX: (925)423-7914 e-mail: tokarz1@llnl.gov
Kambiz Salari	SNL	Tel: (505)844-9836 FAX: (505)844-4523 e-mail: ksalari@sandia.gov
Walt Gutierrez	SNL	Tel: (505)844-5975 FAX: (505)844-4523 e-mail: wtgutie@sandia.gov
Tony Leonard	Caltech	Tel: (606)395-4465 FAX: (606)449-2677 e-mail: tony@galcit.caltech.edu
Karlin Roth	NASA Ames	Tel: (650)604-6678 FAX: (650)604-2238 e-mail: kroth@mail.avc.nasa.gov
Rose McCallen	LLNL	Tel: (925)423-0958 FAX: (925)422-3389 e-mail: mccallen1@llnl.gov
Richard Wares	DOE/HVST	Tel: (202)586-8031 FAX: (202)586-1600 e-mail: Richard.Wares@ee.doe.gov
Fred Browand	USC	Tel: (213)740-5359 FAX: (213)740-7774 e-mail: browand@spock.usc.edu
Walt Rutledge	SNL	Tel: (505)844-6548 FAX: (505)844-4523 e-mail: whrutle@sandia.gov
Jules Routbort	Argonne Nat. Lab.	Tel: (630)252-5065 FAX: (630)252-3604 e-mail: routbort@anl.gov

Department of Energy

Aerodynamic Design of Heavy Vehicles

Overview of Project Plan and Budget

Rose McCallen, Ph.D.

Lawrence Livermore National Laboratory, Livermore, CA

August 1998

University of California Lawrence Livermore National Laboratory

Sandia National Laboratories

USC UNIVERSITY OF SOUTHERN CALIFORNIA

Caltech California Institute of Technology

National Aeronautics & Space Administration

The truck industry relies on wind tunnel and field experiments for aerodynamic design and analysis.

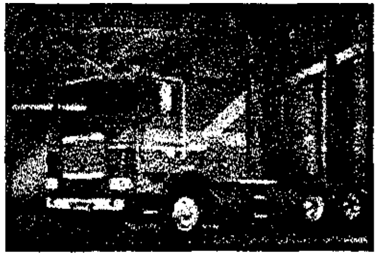
Wind Tunnel Testing

Costly detailed models

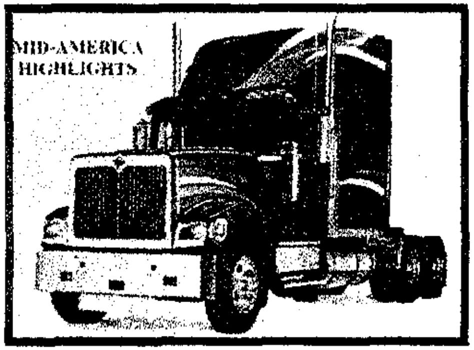
$2,000 to $4,000/hr

Trial-error approach to determine the drag effects due to

- **general tractor shape, under-body and underhood flow**
- **positioning and shaping of head lamps or turning lights**
- **mirror and grab handle configurations and positioning**
- **tractor-trailer gaps and height mismatch**



Cabover Engine



Conventional

Field Testing

Performed by both manufacturer and fleet operators

Issues

A tractor is paired with several different trailers

Almost no aero design interaction between tractor and trailer manufacturers

The effects of design changes on drag are not well understood and computational guidance is needed and welcomed

The MYPP is based on industry needs and consideration of current technology, funding, and DOE interests.

DOE and National Laboratory interest

- **Reduce heavy vehicle drag -> reduce fuel consumption and emissions**
- **R&D for DOE programs**

Industry needs

- **Advanced computational tools and experimental methods**
  - **Understand the effects of design changes**
  - **Simulate fully-integrated tractor-trailers**
- **Design improvements for drag reduction**

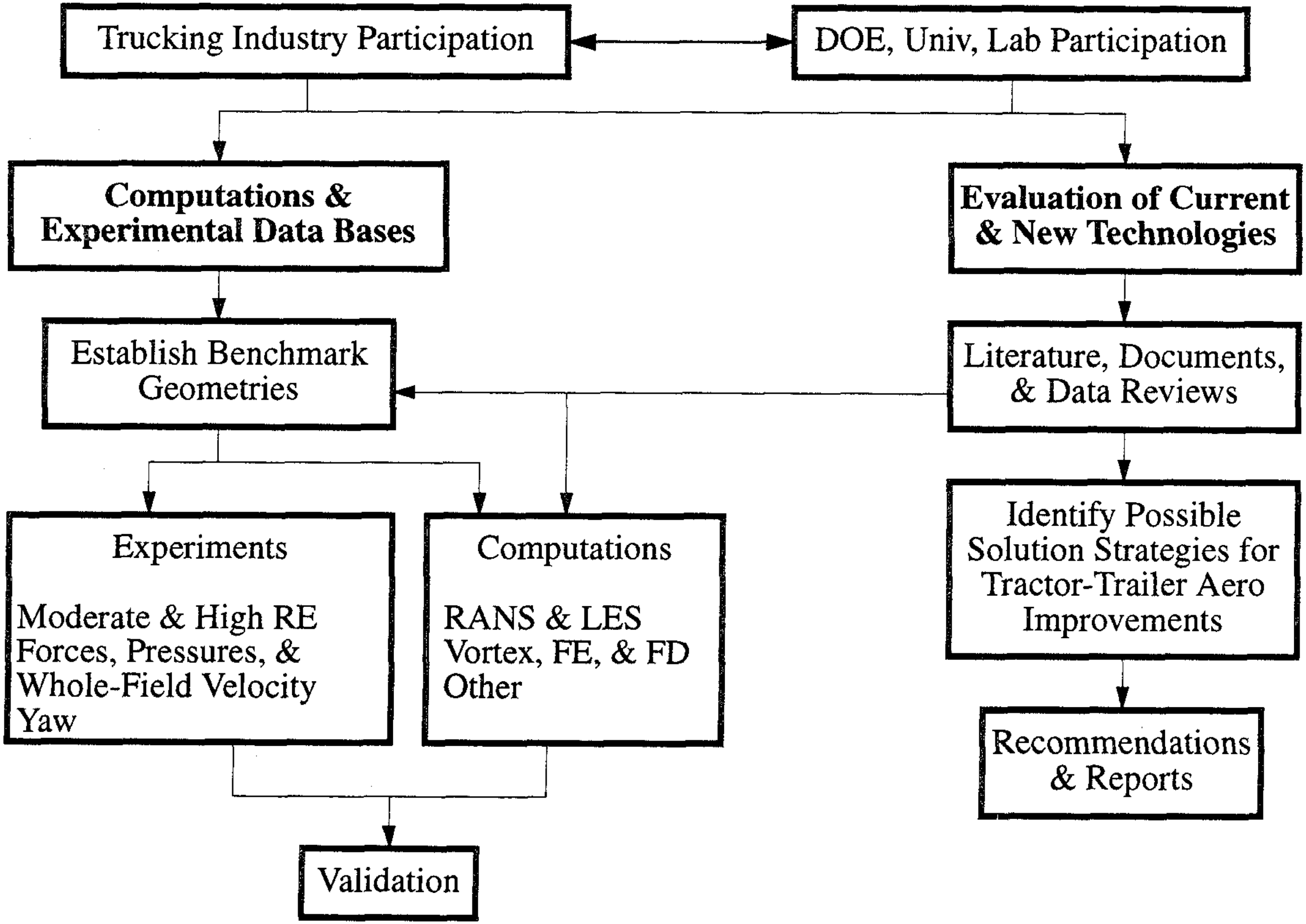
Current technology - CFD is hard!

- **Reynolds-averaged Navier Stokes (RANS) is common approach**
- **Large-eddy simulation (LES) is in development**
- **DPIV measurements can provide full velocity field measurements**

Funding is minimal and we need a plan with a 'near-term impact'

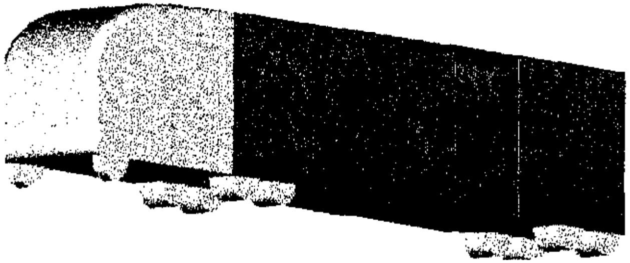
- **$400 K for FY99**

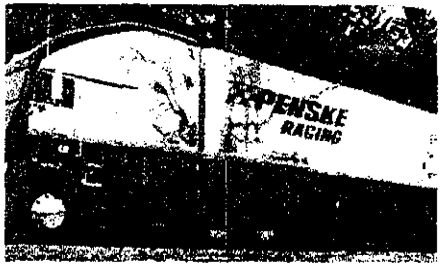
The MYPP focuses on development and demonstration of a simulation capability.



Near-Term Impact: Comparison of RANS and LES and detailed experimental verification for a real truck problem.

Sandia's Model





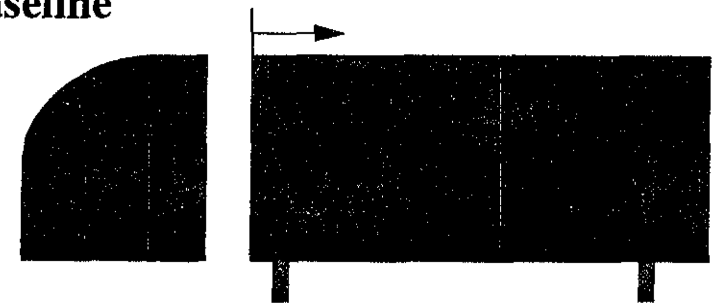
Advantages

Simple geometry with some existing data and some modeling already done

The final detail results will be available for comparison to commercial tools

baseline

height mismatch



gap

Each organization's contributions are critical to the project's success.

Computational Modeling

Rose McCallen (PI)

University of California
Lawrence Livermore National Laboratory

Large-Eddy Simulation using Finite Element Methods

Anthony Leonard

Caltech
California Institute of Technology

Large-Eddy Simulation using Vortex Methods

Don McBride
Walt Rutledge

Sandia National Laboratories

Reynolds-Averaged Modeling using Finite Difference Methods

Experimental Modeling

Don McBride
Walt Rutledge

Sandia National Laboratories

GTS Experiments at Texas A&M

Fred Browand

USC University of Southern California

Moderate Speed Experiments in Wind Tunnel

Jim Ross

National Aeronautics & Space Administration

High Speed Experiments in 7'x10' and 12' Wind Tunnels

Our near-term tasks have been identified and prioritized.

Benchmarks

1. Sandia Body

   Experiments

   - Texas A&M, Re = 1,600,000
   - NASA 7'x10', Re = 1,600,000 and other moderate to lowest Re

     Oil film interferometry, particle image velocimetry, doppler global velocimetry

     Upstream mean velocity profile provided

     0, 5, and 10 degree yaw conditions
   - USC wind tunnel, two Re conditions within 200,000 < Re < 400,000

     With and without trailer/tractor height mismatch and gap

   Computations

   - RANS for high and low Re (SNL)
   - LES for low Re with some attempt at high Re (LLNL and Caltech)
2. New Model Design (USC)
3. Gene's Model for Re sensitivity study (i.e., how high is enough and drag delta's for components)
   - NASA 12', Re_{max} = 5,000,000, model with and without components

Our budget is not consistent with projected funding.

FY99 budget : $400K

	Computations & Experiments	Evaluation of Current & New Technologies	Final Report	Total/Year
FY98	$276K	$34K		$310K
FY99	$630K	$5K		**$635K**
FY99 (low)	($555K)	($5K)		($560K)
FY00	$1,045K	$188K		$1,233K
FY00 (low)	($635K)	($68K)		($703K)
FY01	$1,095K	$188K		$1,283K
FY02	$855K	$161K		$1016K
FY03	$818K	$161K		$979K
FY04	$120K	$124K	$34K	$278K
TOTAL				$5,734K

It was necessary to leverage other funding sources.

SNL	- past data obtained at Texas A&M	Free
	- loan of model to NASA	Free
	- LES R&D	LDRD
	- computational resources	ASCI
USC	- instrumentation	Caltrans, NSF
Caltech	- LES model development	ASCI, DOD
	- computational resources	ASCI, NSF, DOD
NASA Ames	- 7'x10' wind tunnel tests	Free
	- 12' wind tunnel tests	1/3 Cost
	- loan of Navistar's model	Free
LLNL	- computational resources	ASCI
	- LES and code development	ASCI/LDRD (?)

The projected milestones are segregated into benchmark cases with advancing levels of complexity.

Projected milestones for first four years of project (FY98 through FY01)

Task	Milestone
Workshop II	2/98
MYPP with projected budget and milestones	5/98
Continued site visits	8/98, 12/98, 12/99, 12/00
Level 1 Benchmarks: Establish generic shapes and outline test cases for investigation of trailer-tractor height and gap mismatch (Demo)	9/98
Test data at moderate Re for Level 1 benchmarks (Demo)	9/99
RANS, LES/FEM, LES/Vortex computations of Level 1 benchmarks at moderate Re (DEMO)	12/99
Test data at high Re for Level 1 benchmarks (Demo)	6/00
RANS, LES/FEM, LES/Vortex computations of Level 1 benchmarks at high Re (DEMO)	12/00
Workshop III: Possible computation contest	11/99
Level 2 Benchmarks: Establish generic shapes	9/99
Test data at moderate and high Re for Level 2 benchmarks	9/01

Aerodynamics Overview of the Ground Transportation Systems (GTS) Project for Heavy Vehicle Drag Reduction (SAE Paper # 960906 SP-1145)

Walter T. Gutierrez, Basil Hassan,
Robert H. Croll, and Walter H. Rutledge
Sandia National Laboratories
Albuquerque, New Mexico

1996 SAE International Congress and Exposition
Cobo Conference/Exhibition Center
Detroit, Michigan
February 29, 1996

Sandia National Laboratories

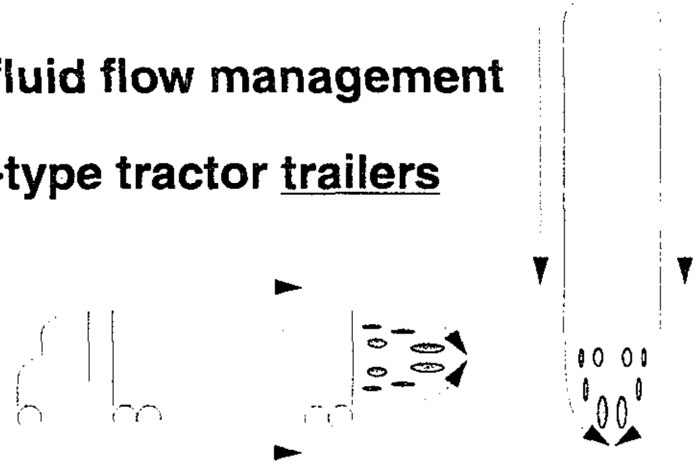
Introduction

Sandia National Laboratories

Engineering Sciences Center

Focus of research

- Increase knowledge level of fluid flow management
- Focus on <u>base</u> region of van-type tractor <u>trailers</u>



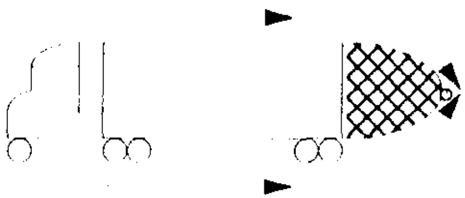
Synergistically use...

Analytical

Computational

Experimental... analysis tools

Draw upon the strengths of each technique

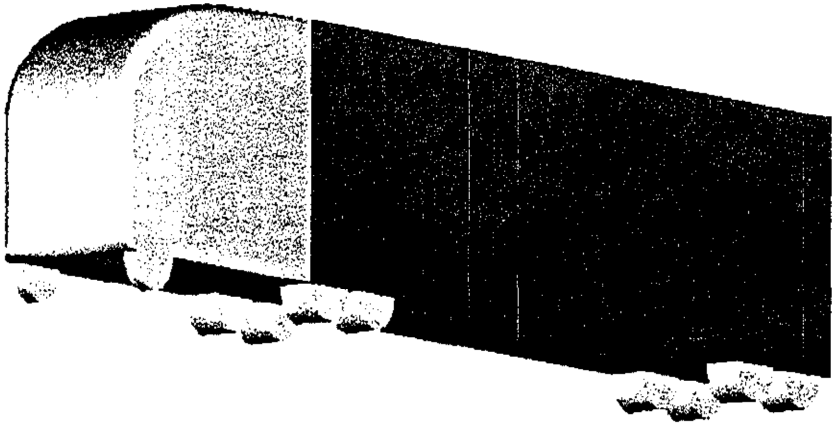


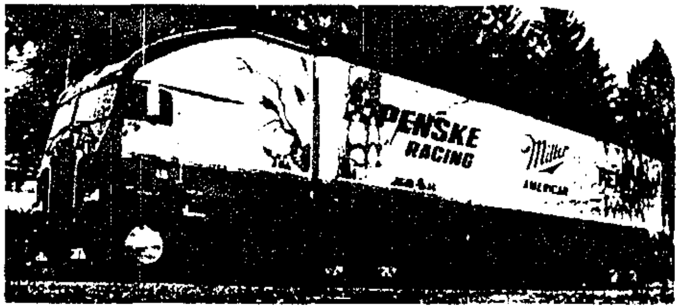
GTS Baseline Geometry

Cab-over tractor trailer

Detail mirrors, wheel wells, tractor-trailer gap not simulated

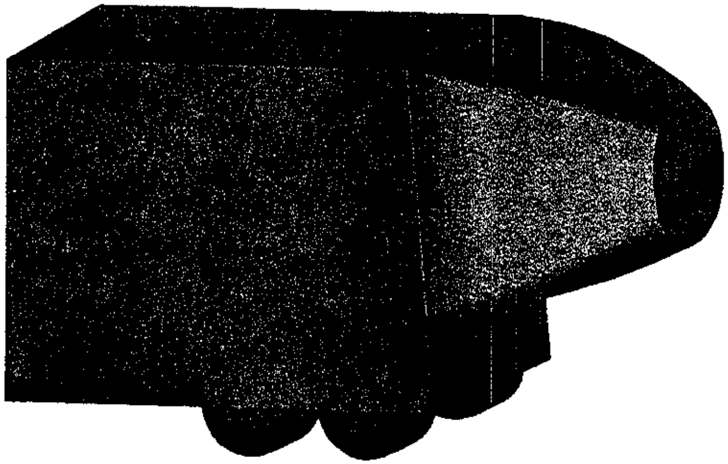
- Simplicity
- CFD grid generation
- Application to general, heavy vehicle transportation industry





Picture Courtesy of
Penske Racing

Add-on Geometries: Ogives and Slants



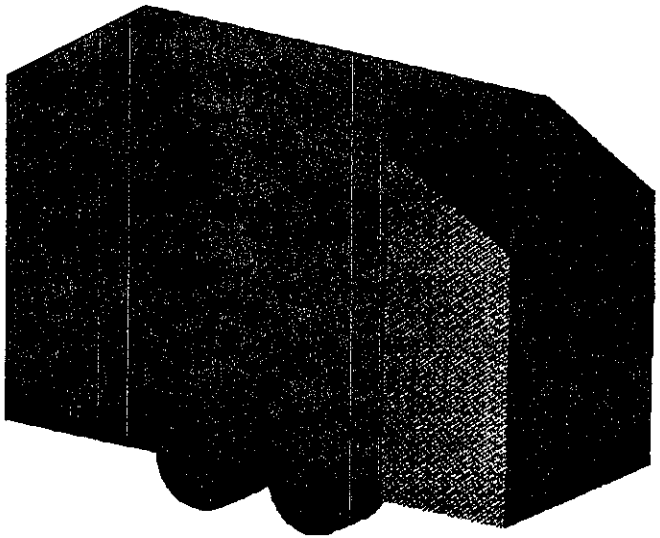
Ogival Boattails

- 1.5 m and 2.4 m long

  "5 ft Ogive" and "8 ft Ogive"
- Tangent at top of trailer and sides
- Blend from square to circle
- Primarily boundary layer separation

Slants

- 5^{o}, 12.5^{o} and 30^{o} fastbacks
- Scaled from work by Ahmed, et al.
- Primarily boundary layer separation and vortex interaction



Experimental Investigation of the Ground Transportation Systems (GTS) Project for Heavy Vehicle Drag Reduction (SAE 960907)

Robert H. Croll, Walter T. Gutierrez, Basil Hassan, Jose E. Suazo, and Anthony J. Riggins
Sandia National Laboratories
Albuquerque, New Mexico

1996 SAE International Congress and Exposition
Cobo Conference/Exhibition Center
Detroit, Michigan
February 29, 1996

Sandia National Laboratories

Experimentation

Purpose: Develop a database on the various GTS geometries for comparison with the concurrent CFD study

Facility

- Texas A&M University Low Speed Wind Tunnel
- Closed circuit with 2.1 m (7 ft) high and 3.0 m (10 ft) wide

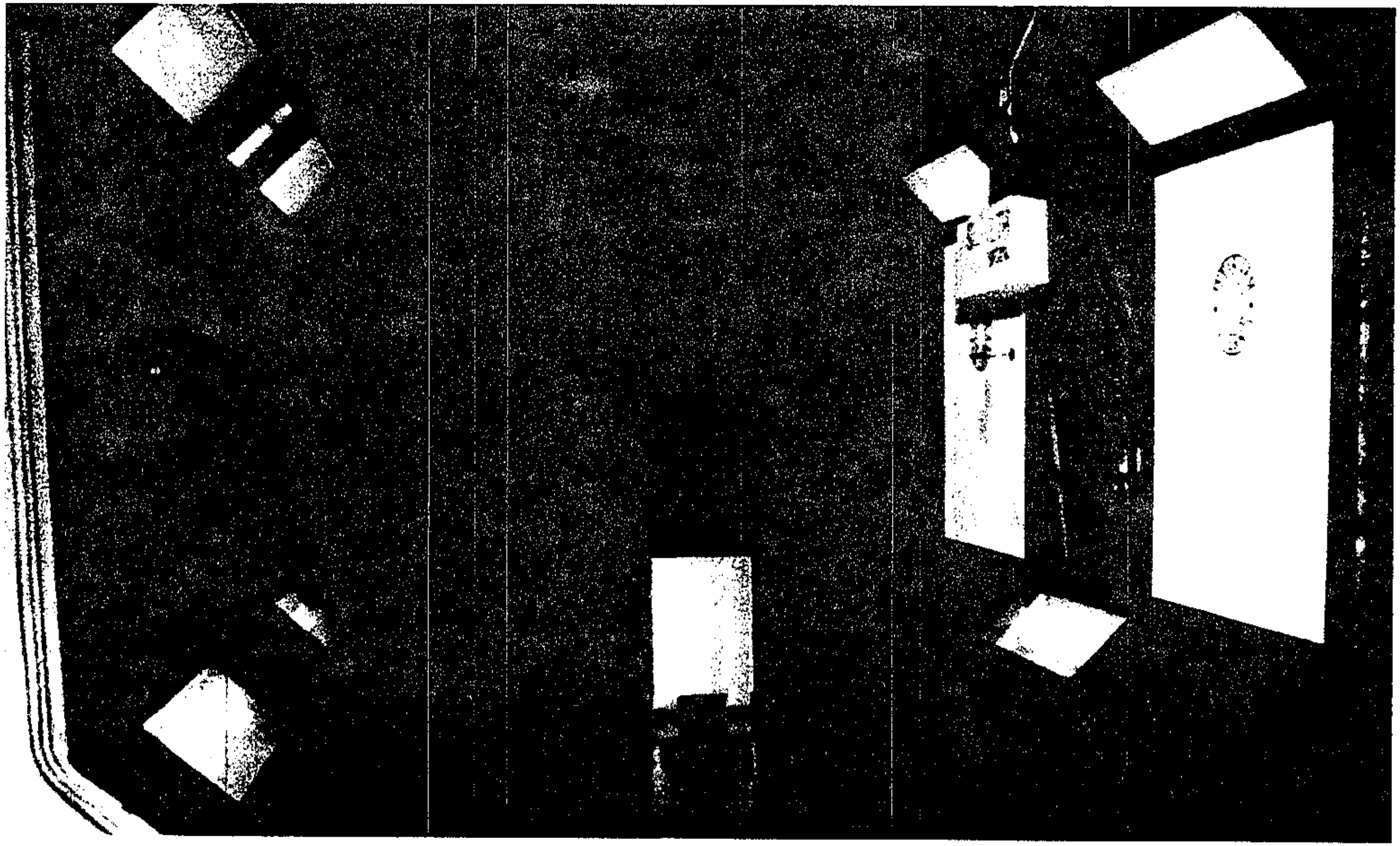
Hardware

- 1:8 scale model
- No boundary layer device
- Baseline with Ogive and Slant add-ons

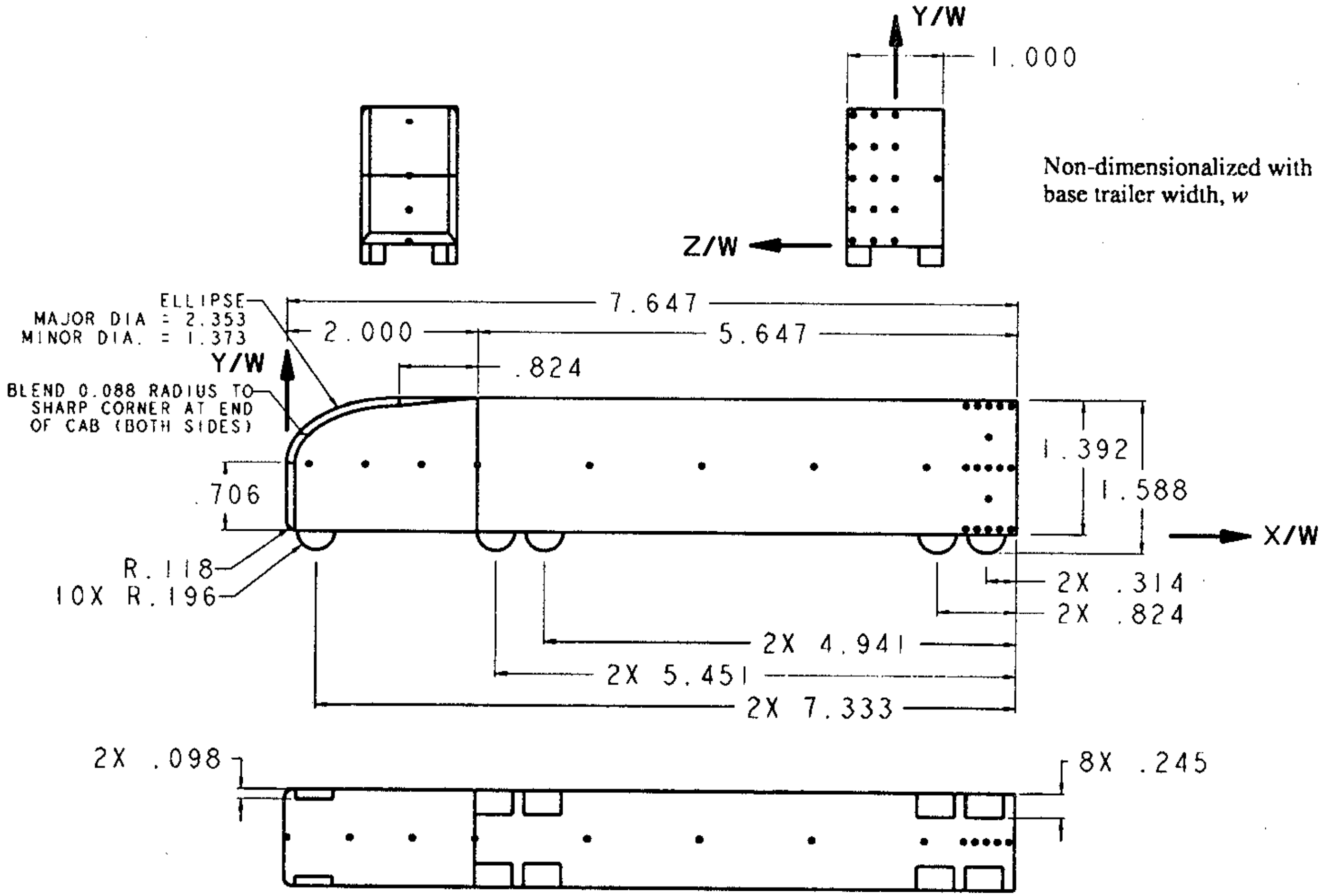
Testing

- Yaw angle range +/-14^{o}
- $Re_W = 1.6 \times 10^6$ (compare to 4.8×10^6 full scale)
- Standard force/moment and wind averaged drag
- Model static surface pressure
- Wake pressure from 7-hole probe
- Smoke, tufts, and tempera paint flow

GTS Baseline Model in Test Section



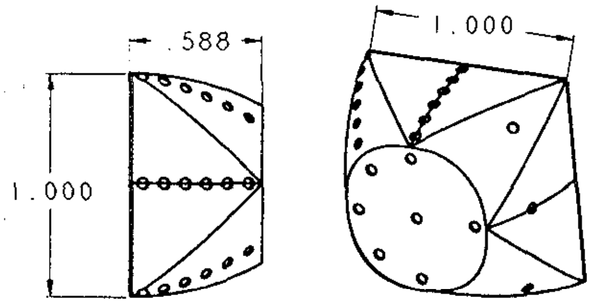
GTS Baseline Geometry Dimensions and Pressure Tap Locations



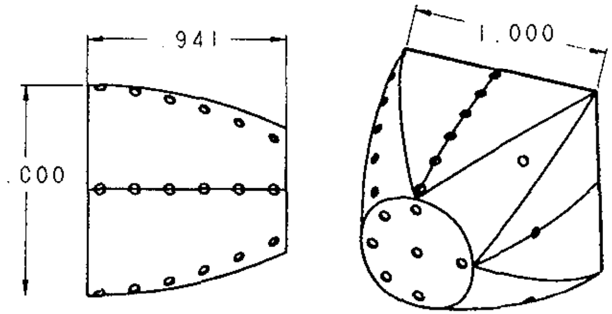
GTS Ogive and Slant Add-on Devices Dimensions and Pressure Tap Locations

Sandia National Laboratories

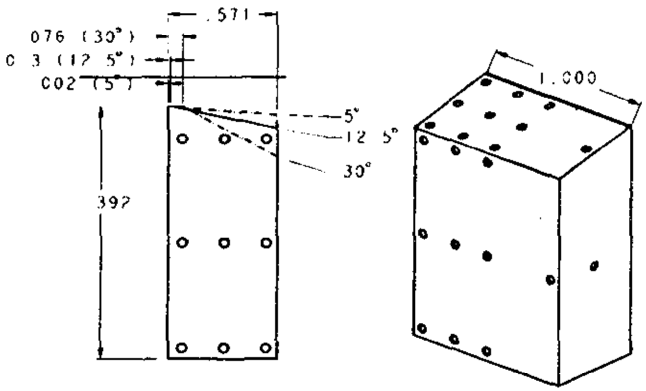
Engineering Sciences Center



5 ft Ogive



8 ft Ogive



Non-dimensionalized with base trailer width, *w*

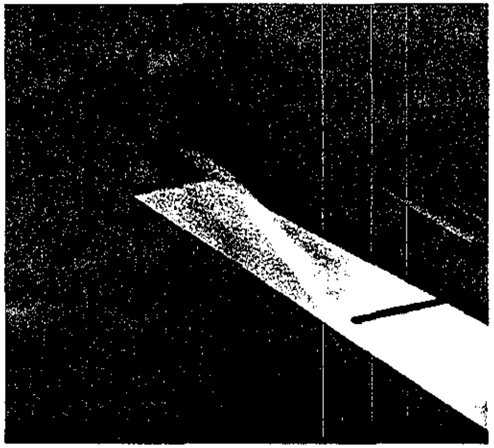
5^{o}, 12.5^{o}, and 30^{o} Slants

Wind Tunnel Test Conditions

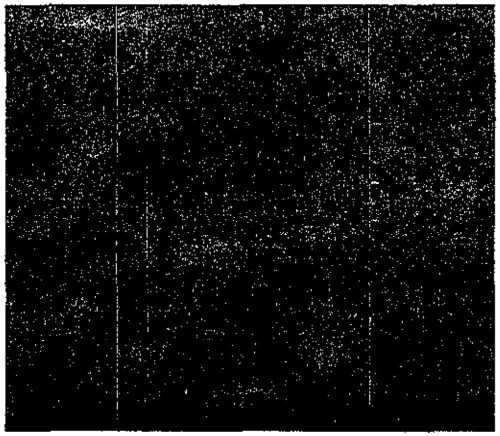
Measurement	Velocity k/hr (ft/sec)	Re_W $(x10^{-6})$	ψ degrees
Force & Moment	285 (260)	1.6	Sweep ±14
Surface Pressure	285 (260)	1.6	Sweep ±14
Wake Pressure	285 (260)	1.6	0, -10
Oil Flow	285 (260)	1.6	0, -5, -10
Body Tufts	216 (197)	1.2	0, ±5, ±10
Wake Tufts	216 (197)	1.2	0, ±5, ±10
Smoke	33 (30)	0.2	0, ±5, ±10

Flow Visualization

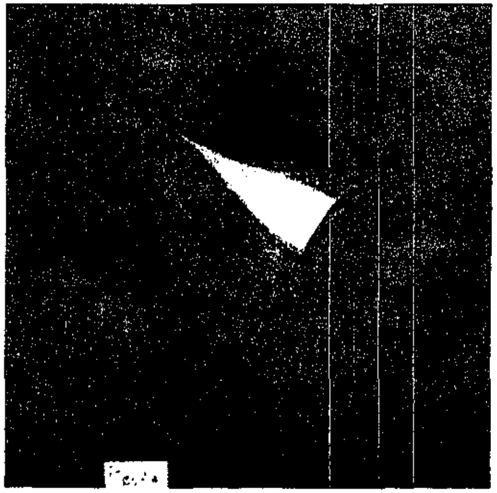
Smoke Flow



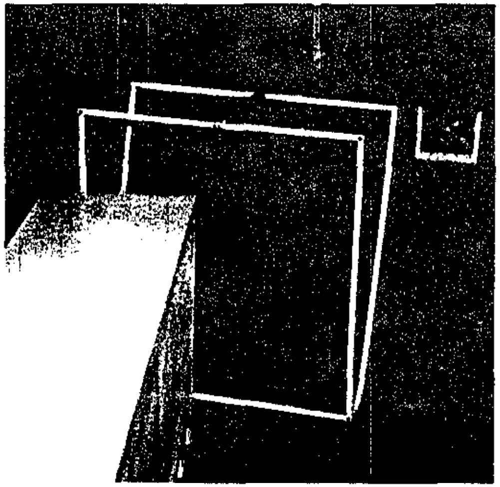
Surface Tufts



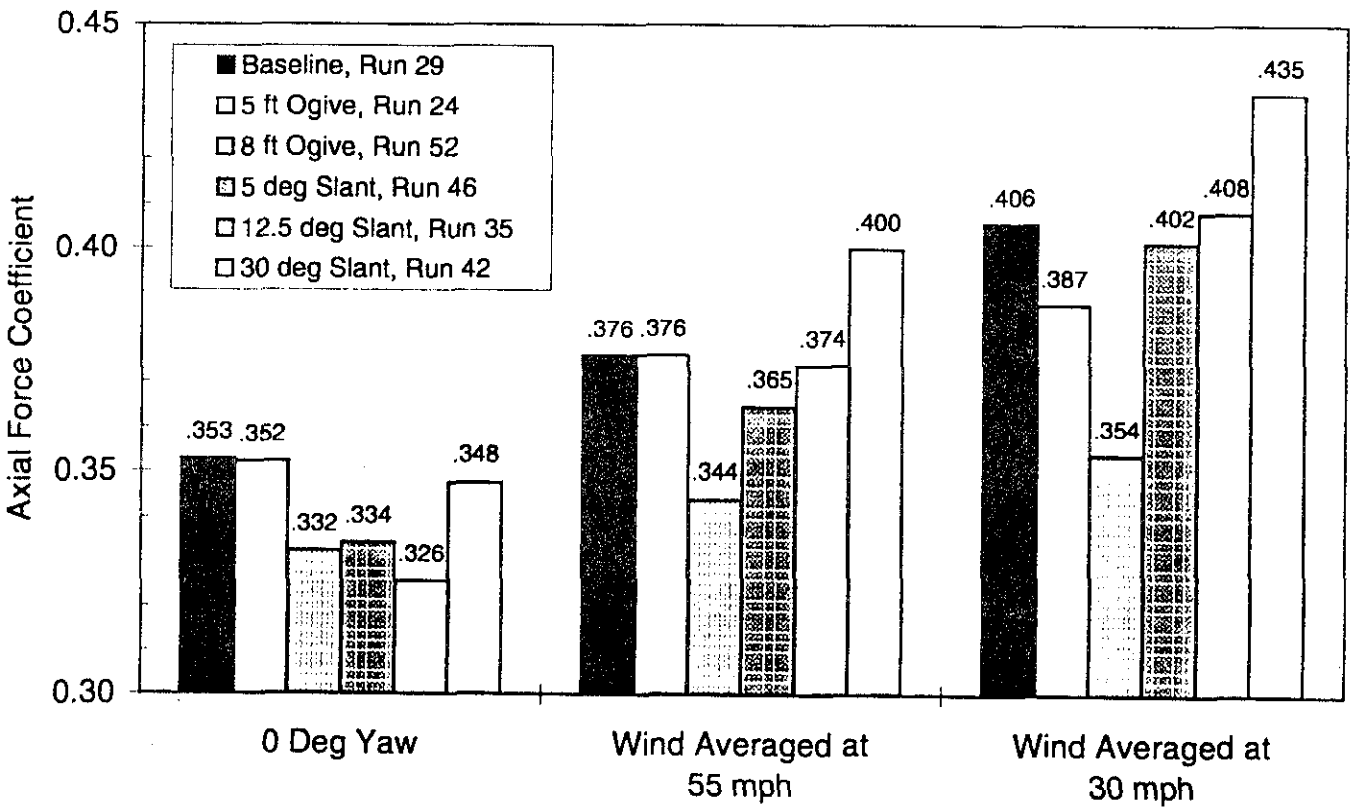
Oil Flow



Wake Tuft Grid



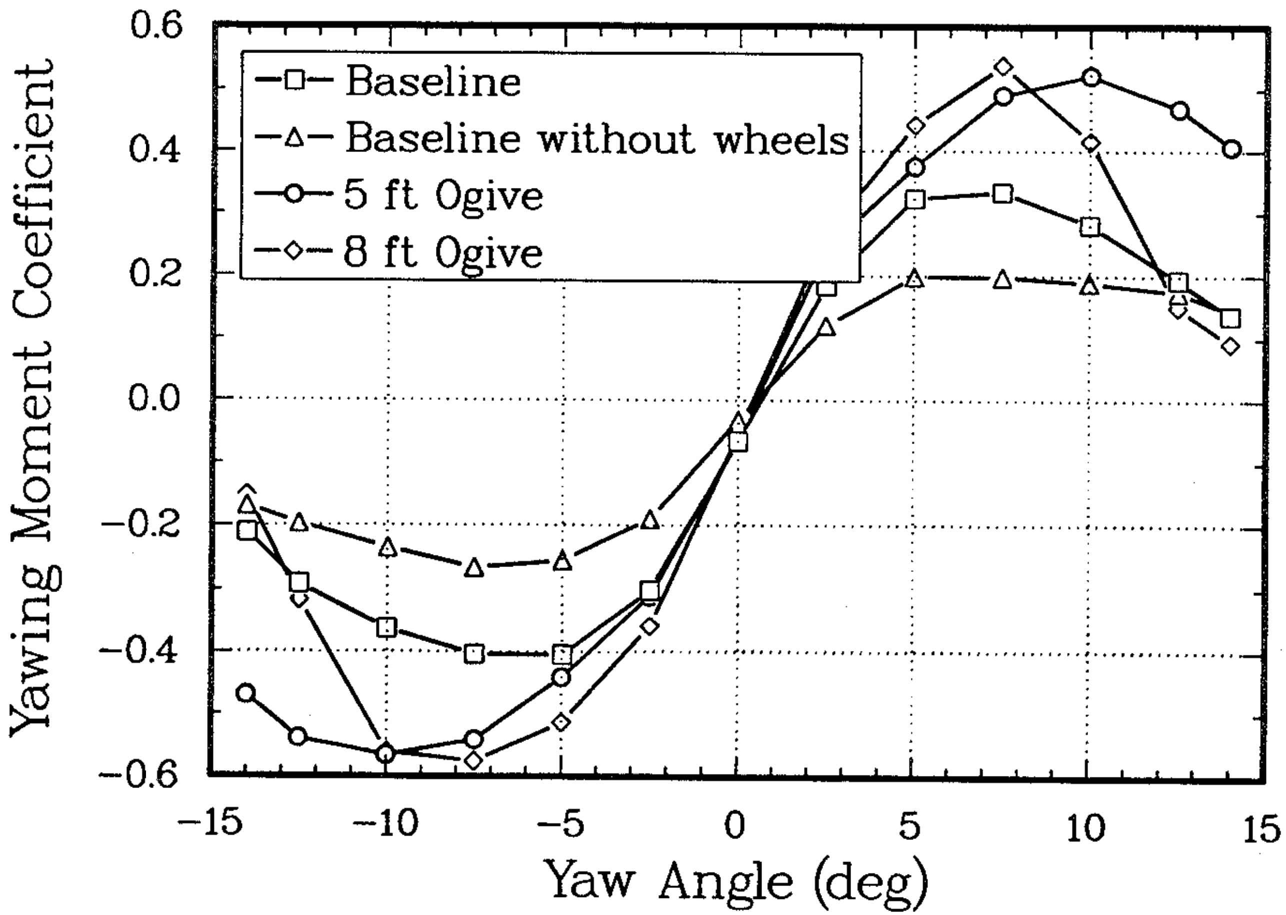
GTS Vehicle
Axial Force ("Drag") Coefficient



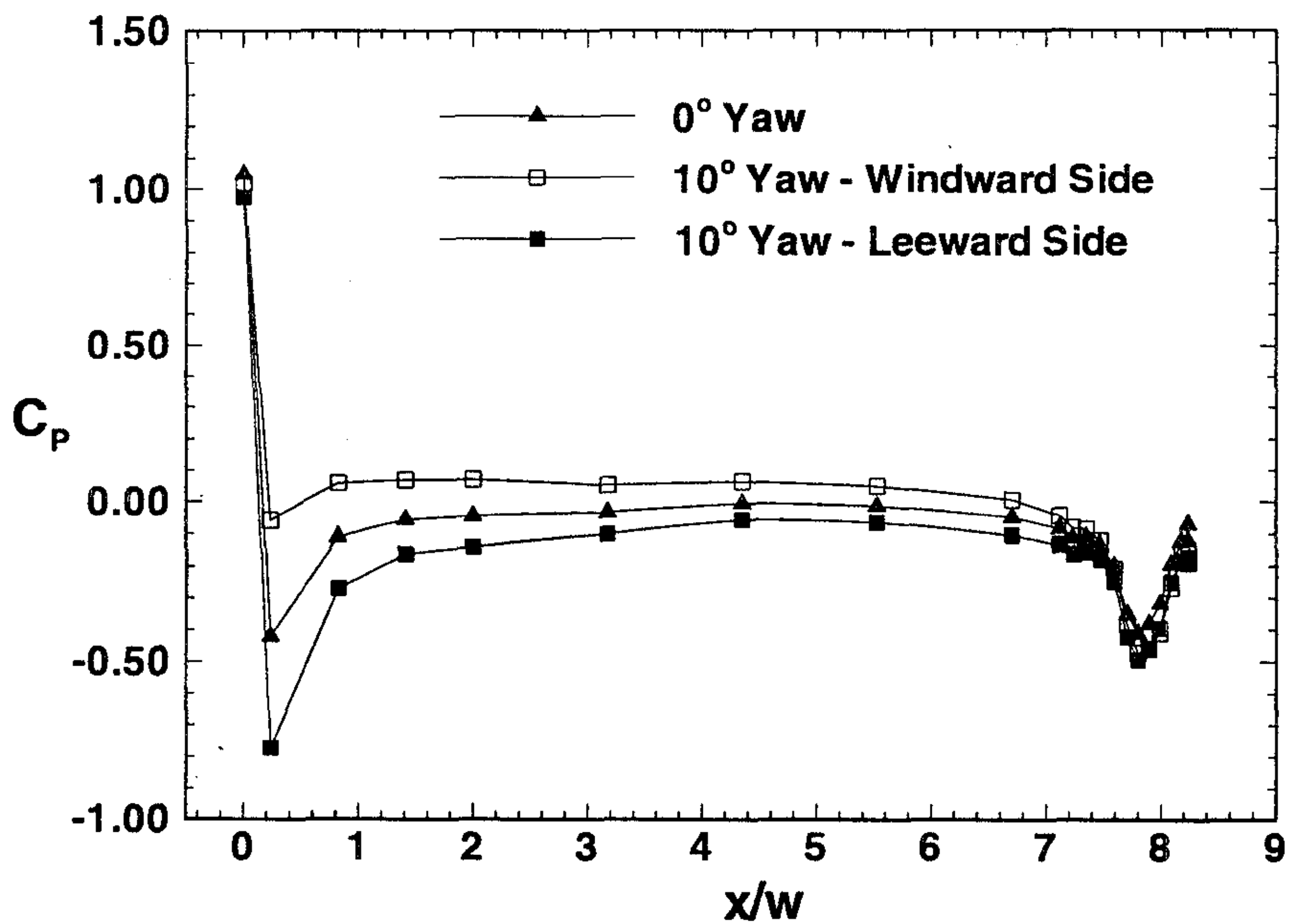
GTS Vehicle
Effects of Ogives on Yawing Moment

Sandia National Laboratories

Engineering Sciences Center



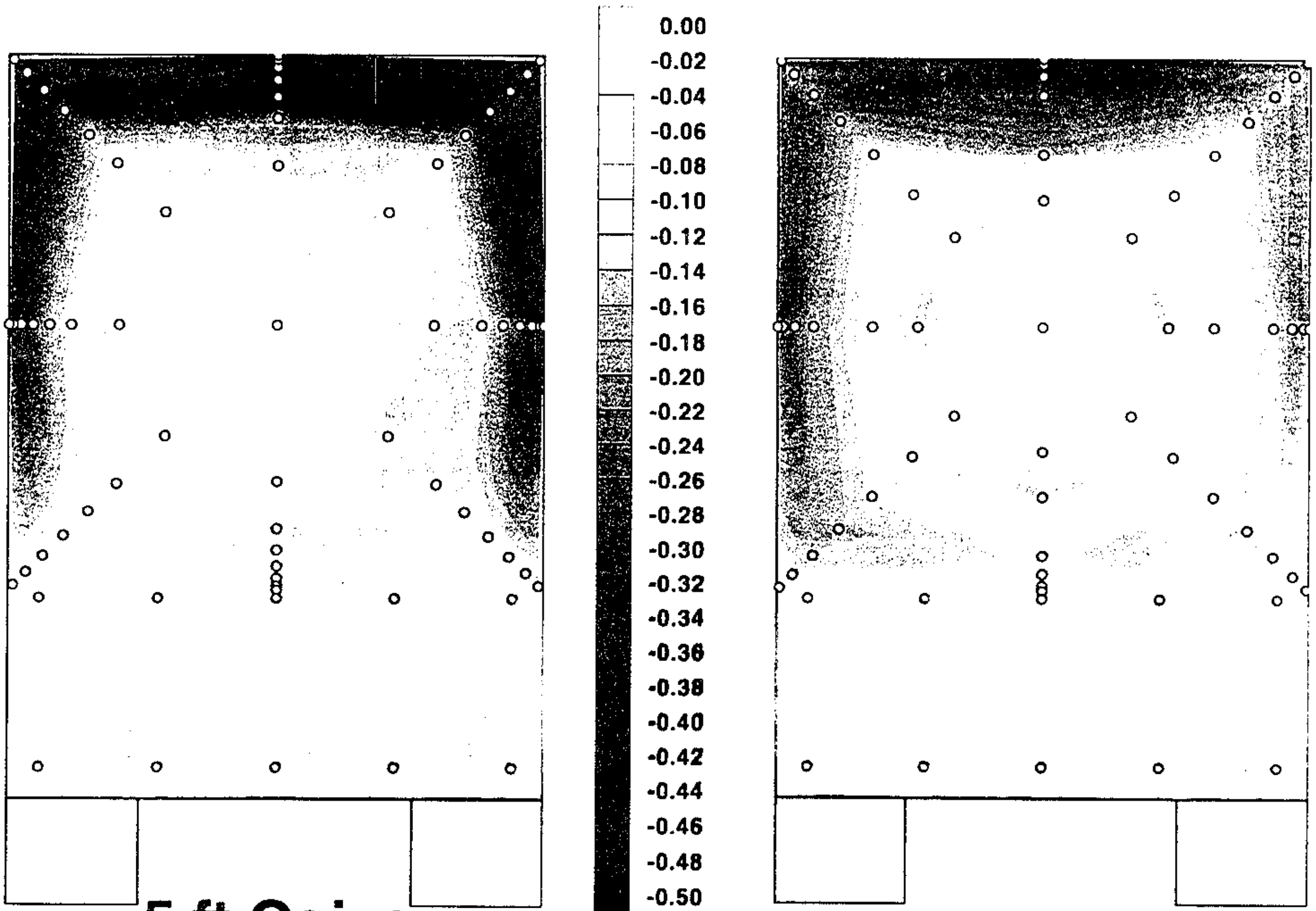
GTS Baseline with 5 ft Ogive
Horizontal Plane Static Pressure



GTS Baseline with Ogives -- 0° Yaw
Base Static Pressure Coefficient

Sandia National Laboratories

Engineering Sciences Center

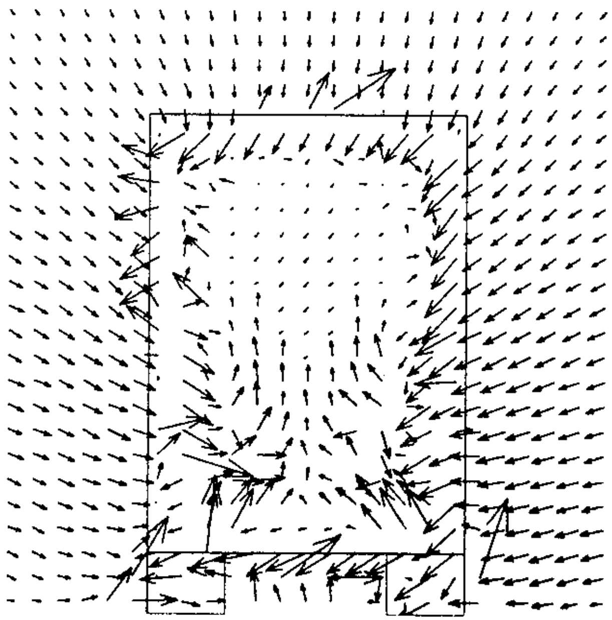


5 ft Ogive **8 ft Ogive**

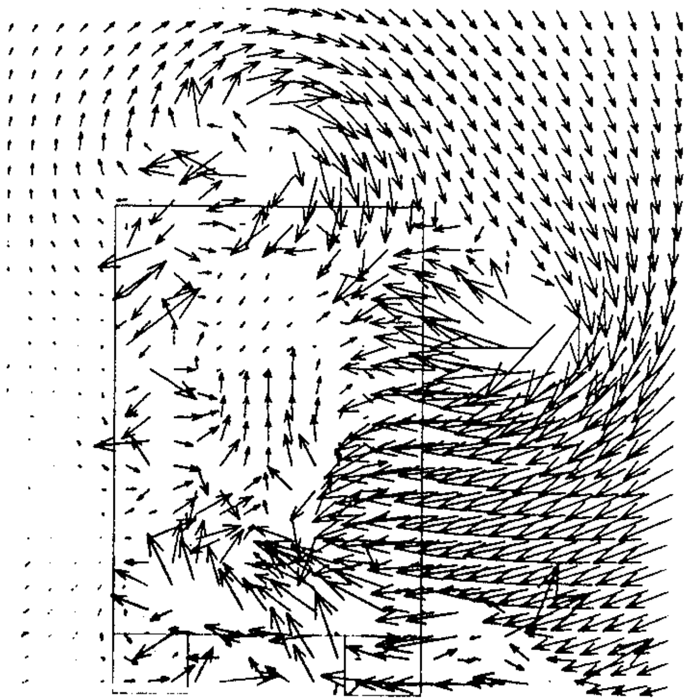
GTS Baseline
Wake Velocity Vectors at Station 2

Sandia National Laboratories

Engineering Sciences Center



0° Yaw



10° Yaw

USC TASKS

Modify the wind tunnel ground plane to accept a circular yaw plate.

- Yaw increments to be stepper-motor controlled
- Continuous yaw increments to ± 12 degrees
- Provision for tractor & trailer to be mounted separately on the yaw plate—with stepper-motor controlled gap

Install new droplet atomizers for particle generation for whole-flow field velocity measurement.

- Purchased a commercially available generator
- Apply smoke in pulse-mode operation

Construct YAG laser light path & optics.

- Horizontal slice and vertical slice viewing

Truck geometries.

- Generate coordinates for simple cab & trailer shapes
- Shapes to be fabricated on 4-axis CNC milling machine

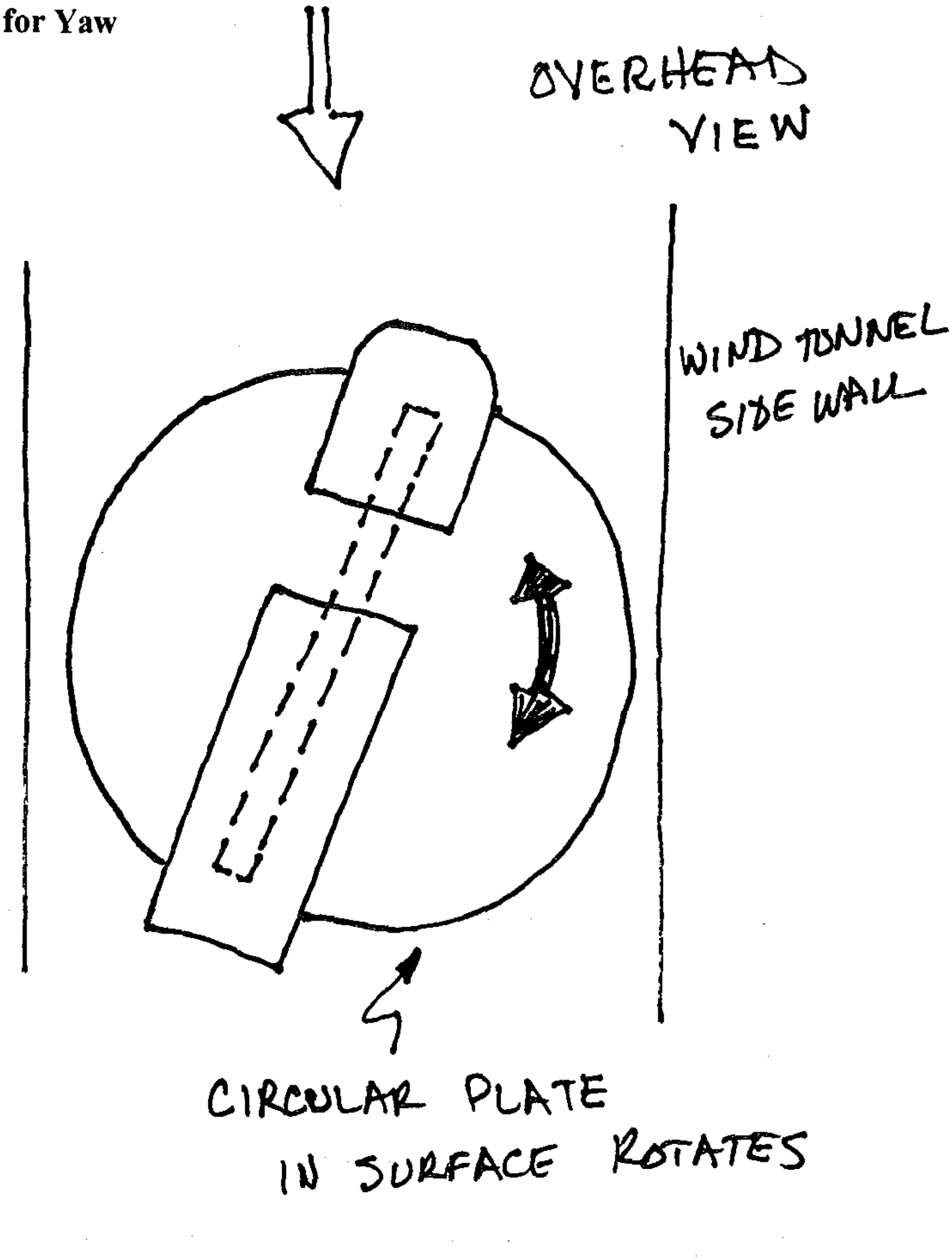
Potential flow calculations.

- Flow over cab using AMES panel code
- Surface pressure distribution
- Identify regions of possible early—and unwanted—separation

Progress in applying whole-flow field (DPIV) measurement.

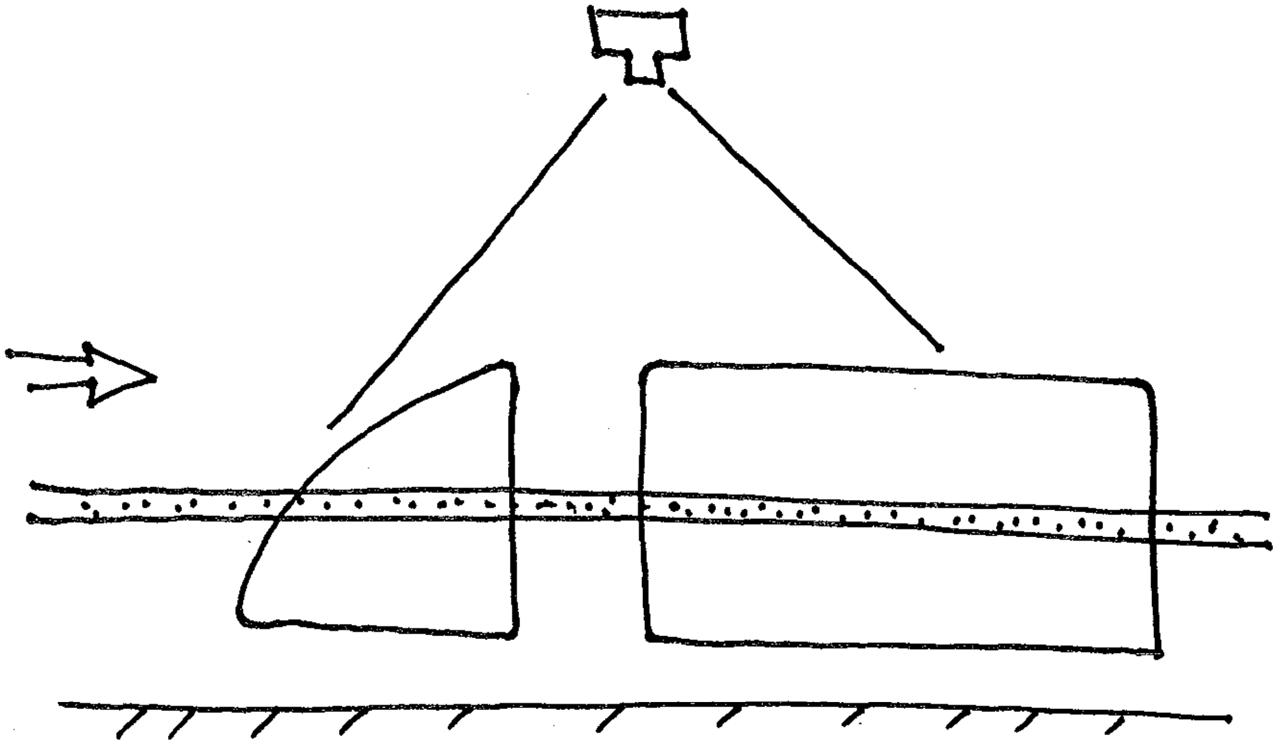
- Back-to-back vehicle geometry as a model for cab-trailer gap

Provision for Yaw



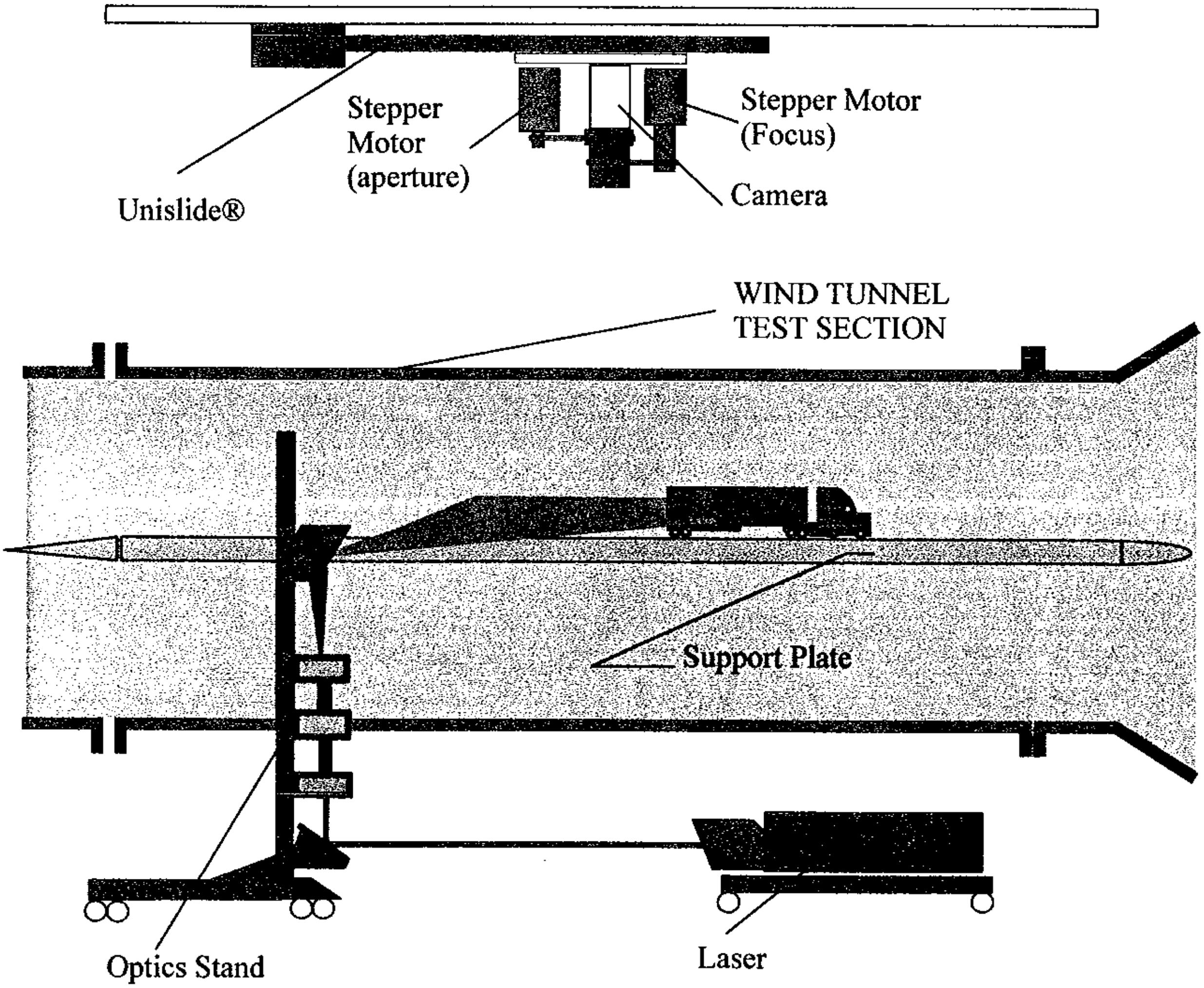
MEASUREMENTS: WHOLE-FIELD VELOCITY

- Flow seeded with small droplets, ≈ 5-25 microns in size
- Laser light sheet forms a plane
- Video camera views normal to the plane, 1000 x 1000 pixels



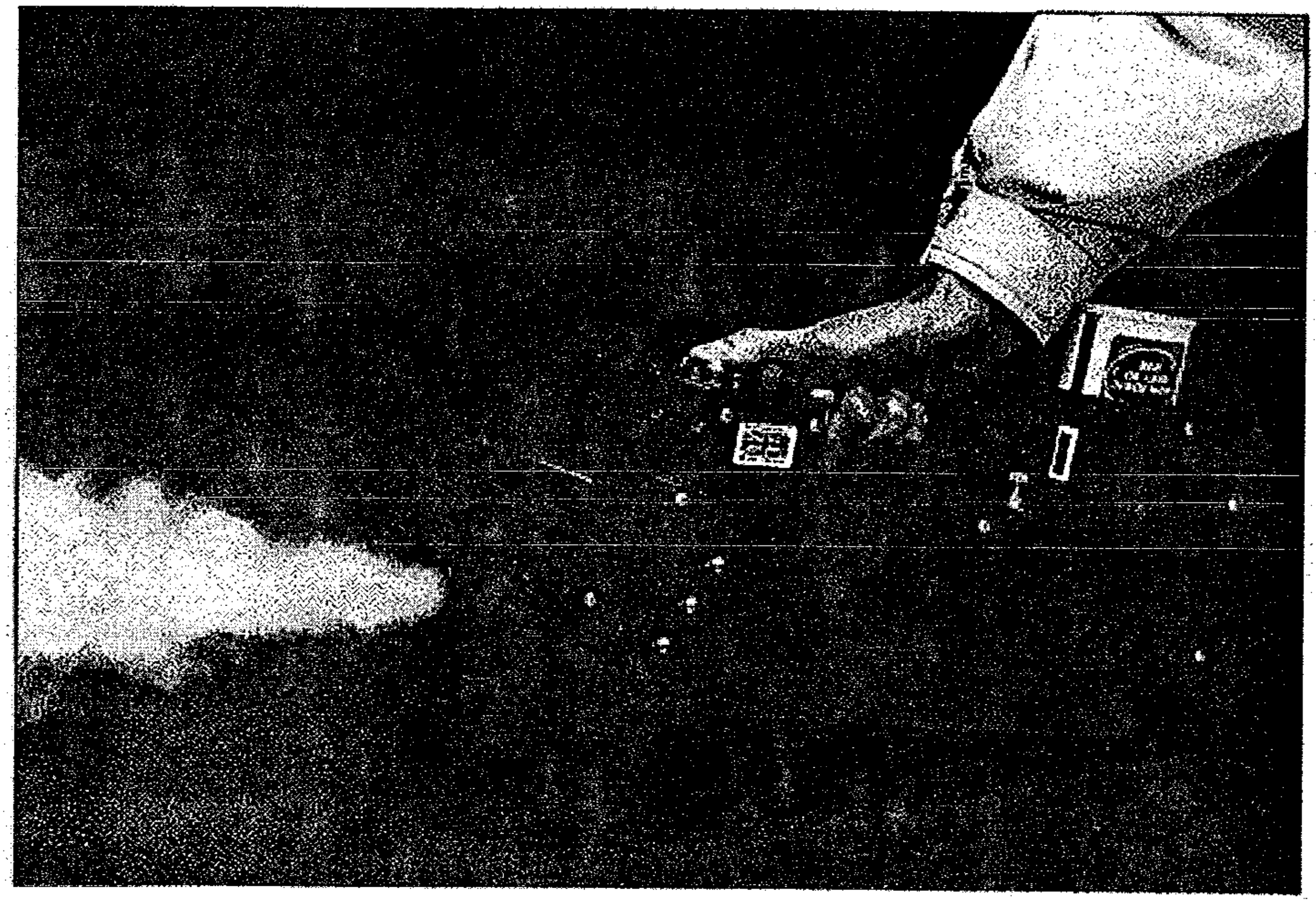
- Turn the laser on for 5-10 nanoseconds, 5-10 x 10^{-9} seconds, and take Picture Number 1
- Wait 20-100 microseconds, 20-100 x 10^{-6} seconds
- Turn another laser on for 5-10 nanoseconds and take Picture Number 2
- Compare Picture 1 and Picture 2, and determine the movement or "flow"

EXPERIMENTAL SETUP

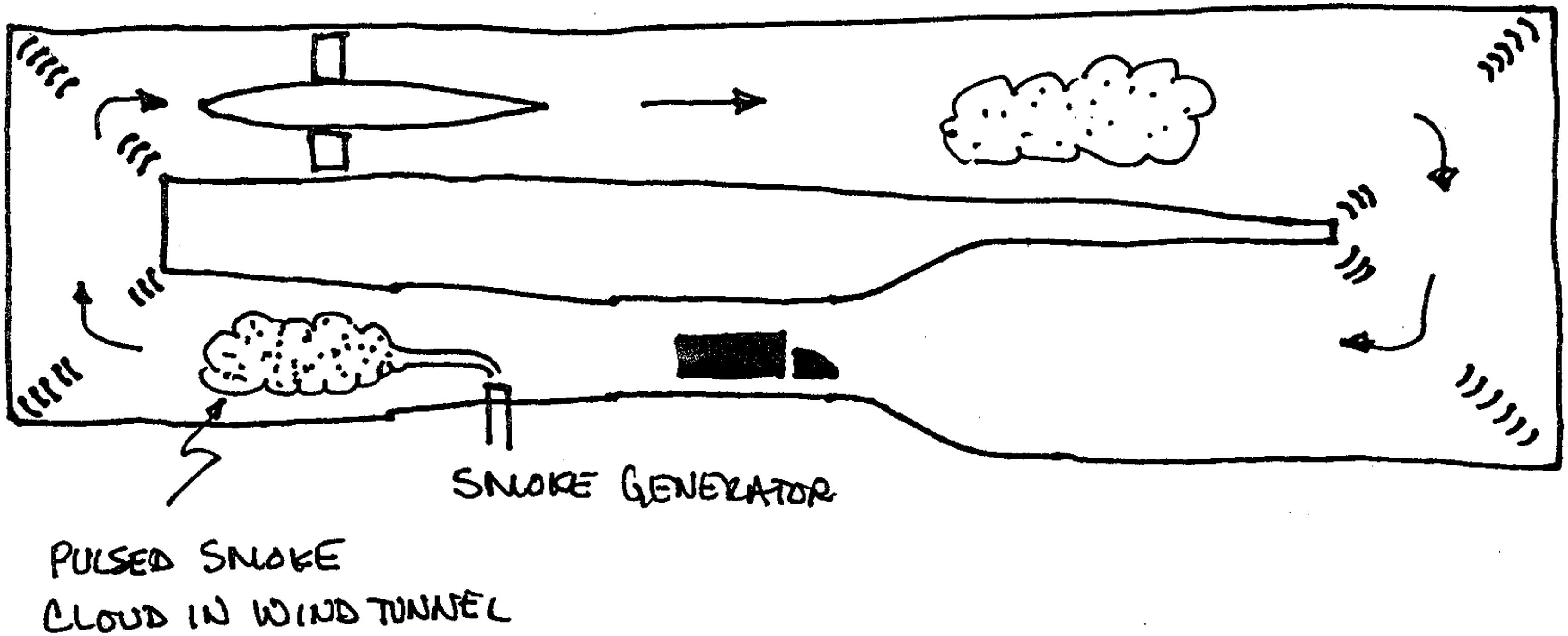


CORONA INTEGRATED TECHNOLOGIES INC.

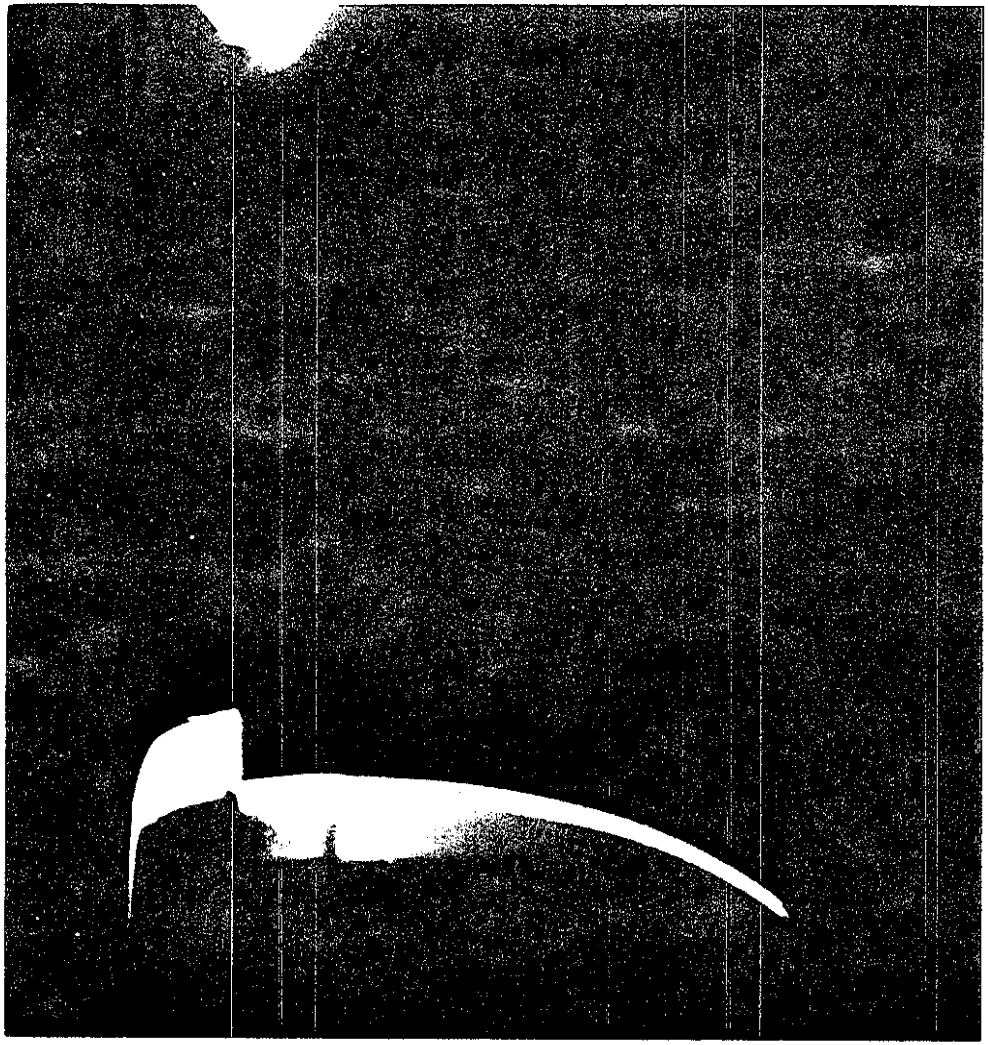
CORONA COLT SMOKE GENERATORS

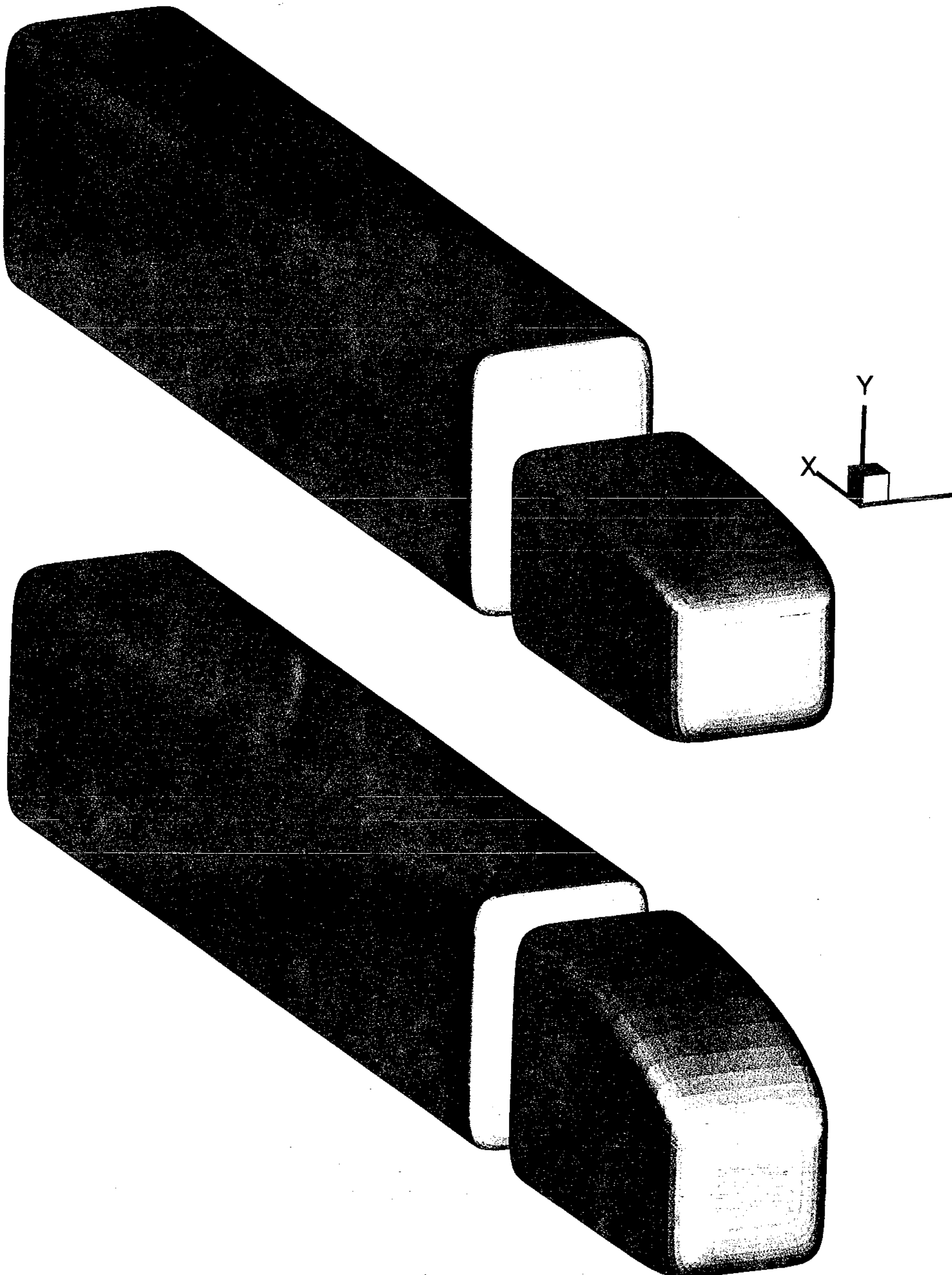


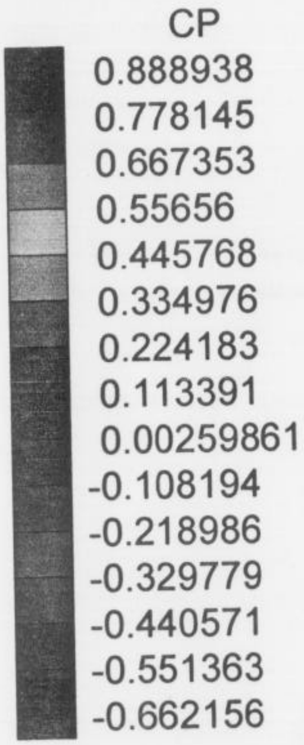
- Property Security
- Air Flow Visualization
- Wind Tunnel Testing
- Flight Crew Training
- Police Force Training
- Air Duct Leak Detection
- Heating/Air Conditioning
- Drain System Leak Testing
- Air Filter Design
- Evacuation Training
- Special Effects
- Smoke Simulation
- Sprinkler System Leak Test
- Chimney Flue Leak Test
- Oxygen System Leak Test
- Mine Shaft Air Test
- Entertainment
- Fumigation

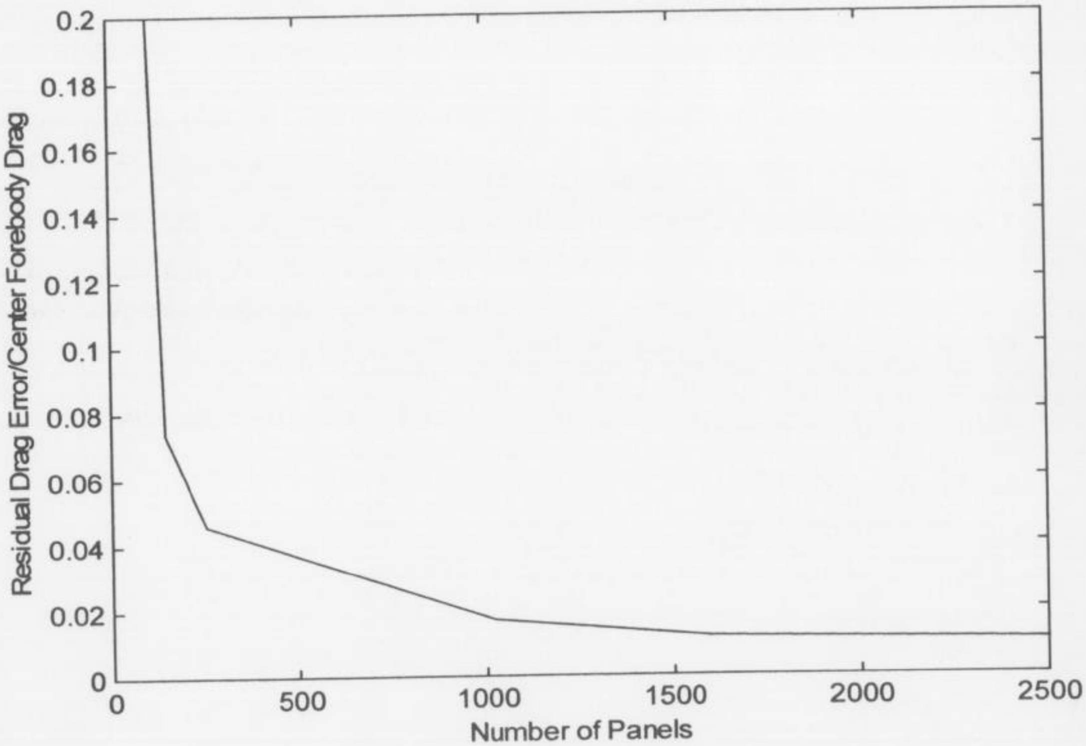


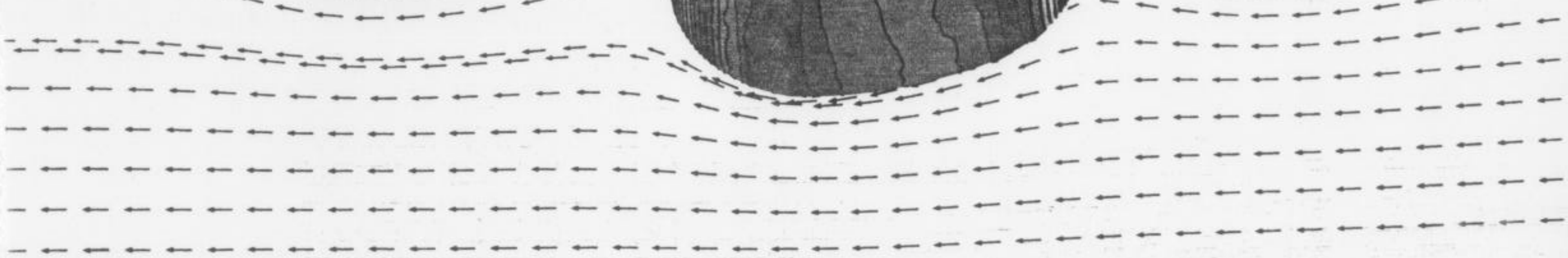
PULSED SMOKE GENERATION

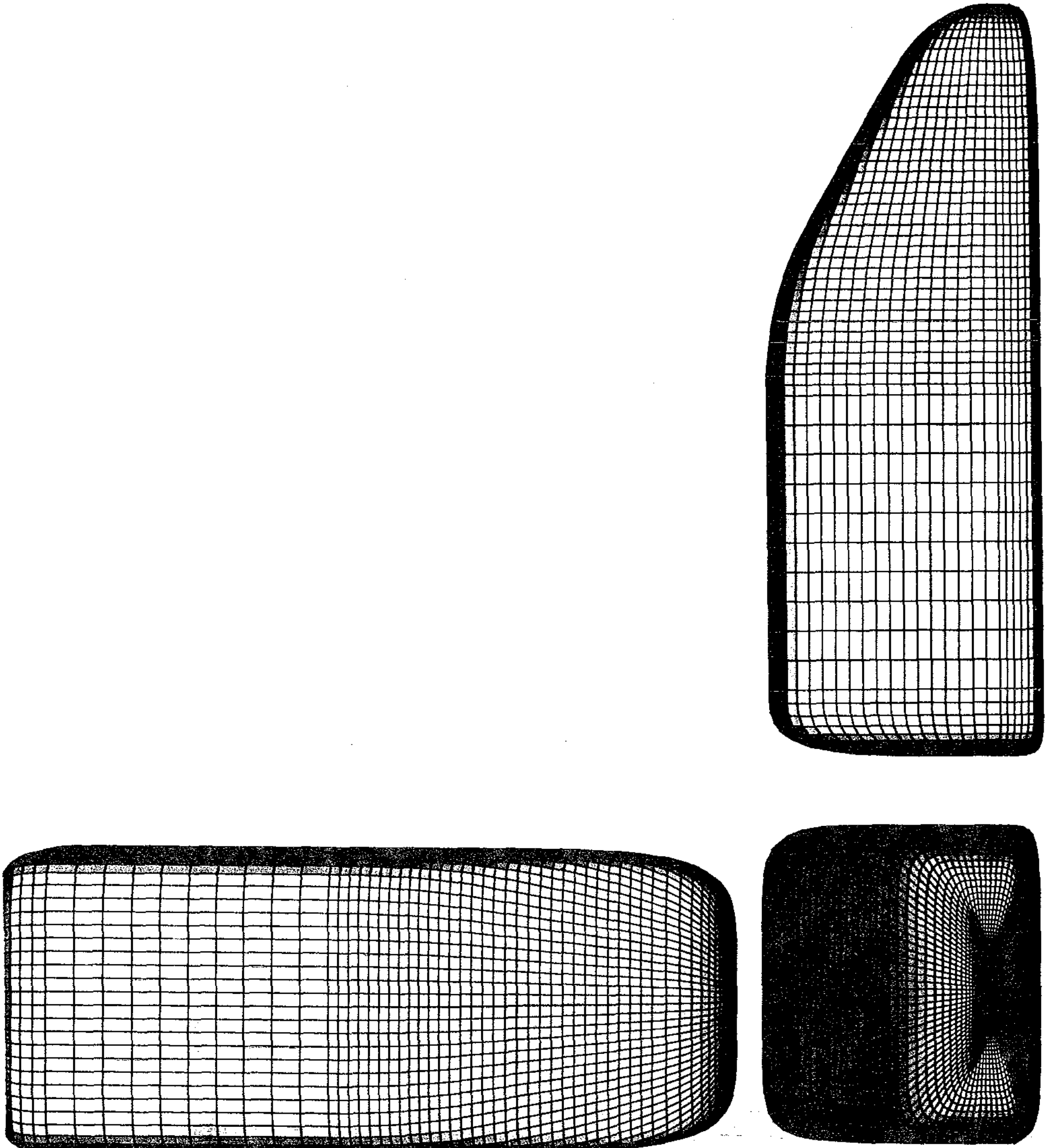



Y
X


CP
0.888938
0.778145
0.667353
0.55656
0.445768
0.334976
0.224183
0.113391
0.00259861
-0.108194
-0.218986
-0.329779
-0.440571
-0.551363
-0.662156


Residual Drag Error/Center Forebody Drag
Number of Panels
0
0.02
0.04
0.06
0.08
0.1
0.12
0.14
0.16
0.18
0.2
0
500
1000
1500
2000
2500





CORONA INTEGRATED TECHNOLOGIES INC.

CORONA COLT SMOKE GENERATORS

The **COLT** Series of Smoke Generators are primarily designed for the rigorous military and industrial marketplace. They are widely used throughout the world by Military Forces, Fire & Police Departments, Health Authorities, Airlines and the Entertainment Industry.

THE SMOKE

- Dry, dense, safe
- Non toxic, non irritant, non contaminant
- Non conductive, non corrosive, non flammable
- Non staining
- Leaves no residue
- Unaffected by adverse temperatures
- Harmless to computers, cameras, electronics and other sensitive equipment and machinery
- Tested by the Canadian Centre for Occupational Health and Safety

THE GENERATOR

- Compact and robust design
- Precision machined, solid steel heater block with removable spiral form core
- Two cartridge heaters providing uniform heating throughout the block
- Heat sensing at the core of the block
- Variable smoke output (zero - maximum)
- Exceptionally easy to use
- Minimal maintenance
- EMC compliance and CE accreditation
- ISO 9001 for design and manufacture

PRINCIPLES OF OPERATION

Corona Smoke Generators produce a thermal fog by introducing a fluid solution into a heater block under pressure. The solution vapourizes as it passes through the heater block. When the vapor is re-introduced into the atmosphere it cools, causing it to condense and form "smoke" particles that are suspended in the air.

The thermal fog particles produced by a Corona machine have a diameter that is one fifth the size of those produced by any other special purpose smoke systems. They hold less than one hundredth the amount of liquid and drop at a rate that is fourteen times slower. Due to its fine mist composition very little fluid is required to create Corona's thermal fog. Corona's unique Smoke Fluid is contained in an air tight canister and pressurized by an inert gas.

The Colt produces a smoke that is dry, dense and long lasting, even after the Smoke Generator has been switched off. The smoke is capable of withstanding temperatures in excess of 65^{0}C. The smoke produces extremely low visibility, which is achieved very quickly and maintained for extended periods of time, making it ideal for Fire Training, Building Evacuation Training, Leak Testing, and Airflow Visualization.

QUALITY CONTROL

Corona generators are designed and manufactured to ISO 9001 standards, the highest level of Quality Control available. All Colt Smoke Generators are Factory Pre-set and tested prior to shipping.

SPECIFICATIONS (approximate)

Weight:		12 lbs.
Size:		19 X 6.5 X 9 (Inches)
Finish:		Epoxy Powder Coat
Power supply:		110V, 60Hz,
Power consumption	**- Colt 4:**	1.1KW
	- Colt 4 Turbo:	2.2KW
Warm up time from cold:		5 minutes
Duration of aerosol at maximum output		18-20 minutes
Smoke output:	**- Colt 4:**	3,400 cu.ft. /min at 4 ft. visibility
	- Colt 4 Turbo:	6,350 cu.ft. /min at 4 ft visibility
Smoke particle size:		0.2 - 0.3 micron

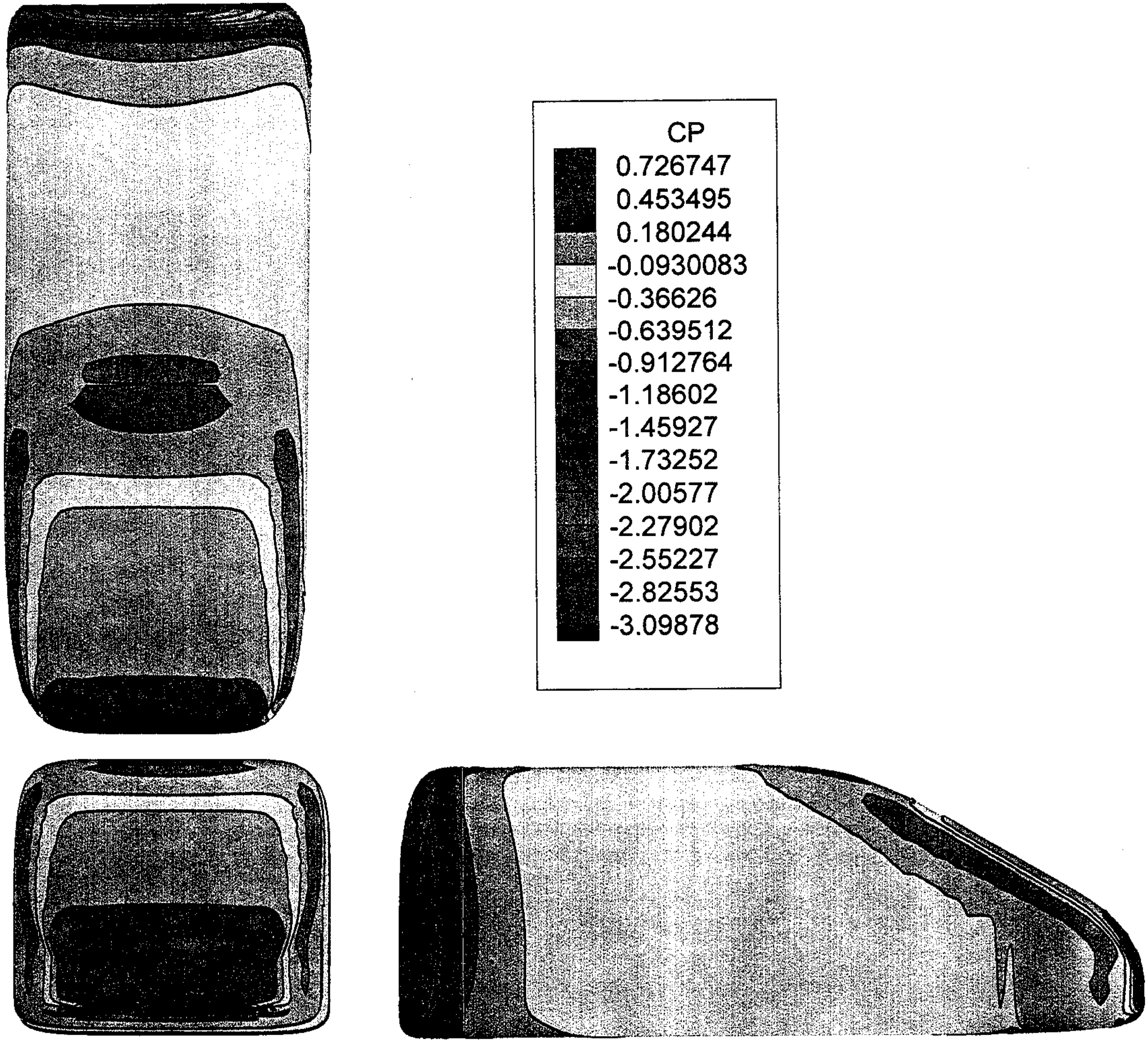
STANDARD EQUIPMENT

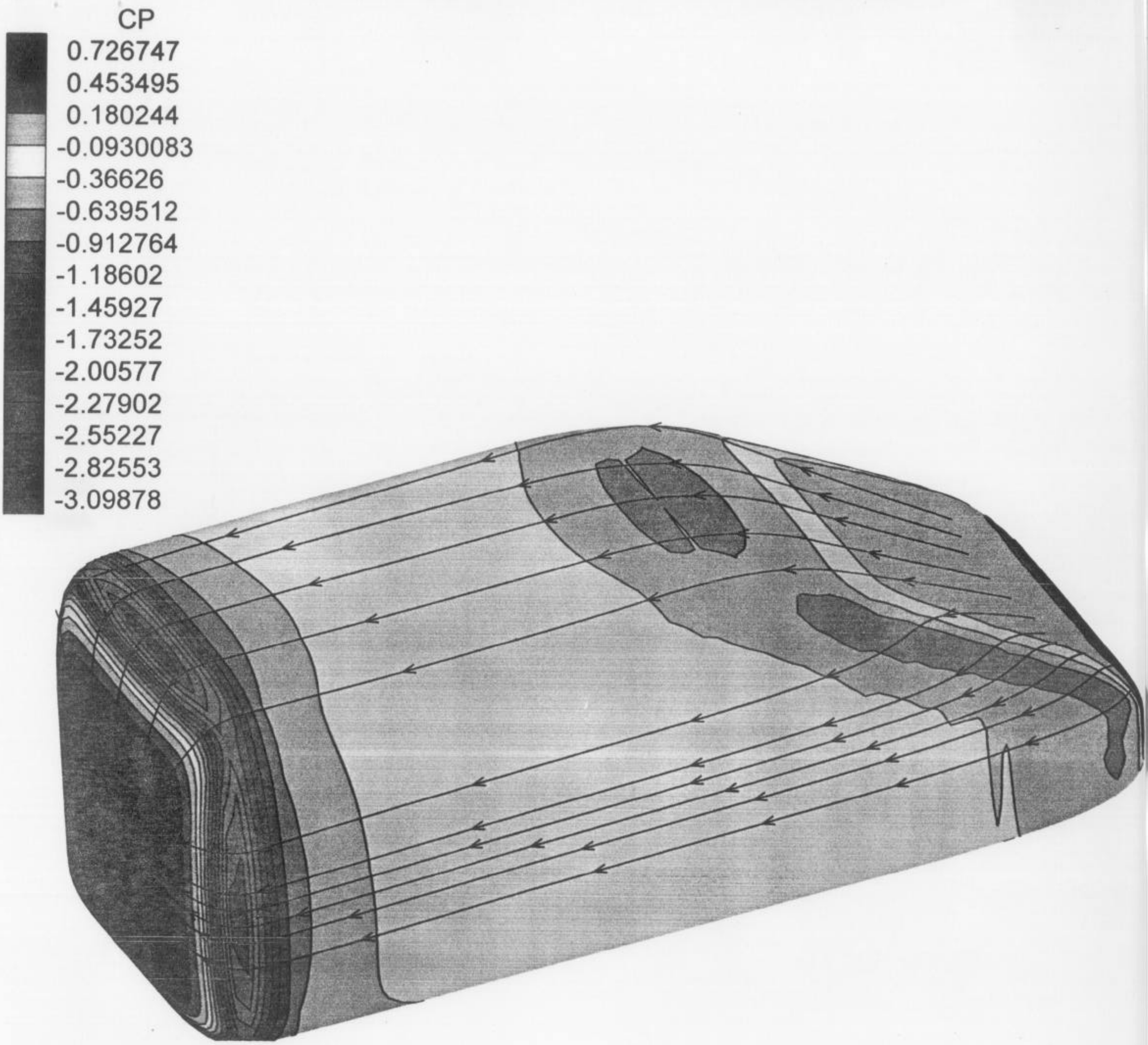
1 Gas Propellant Canister
Operating Instructions
Service Kit

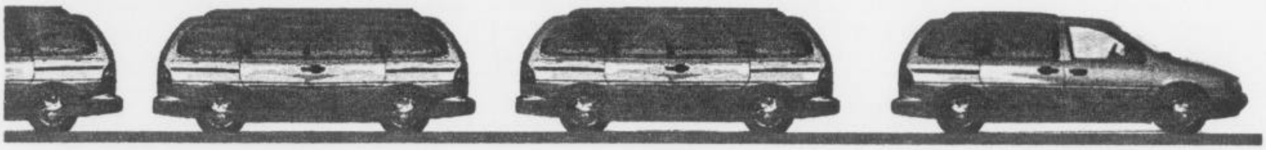
OPTIONAL EXTRAS

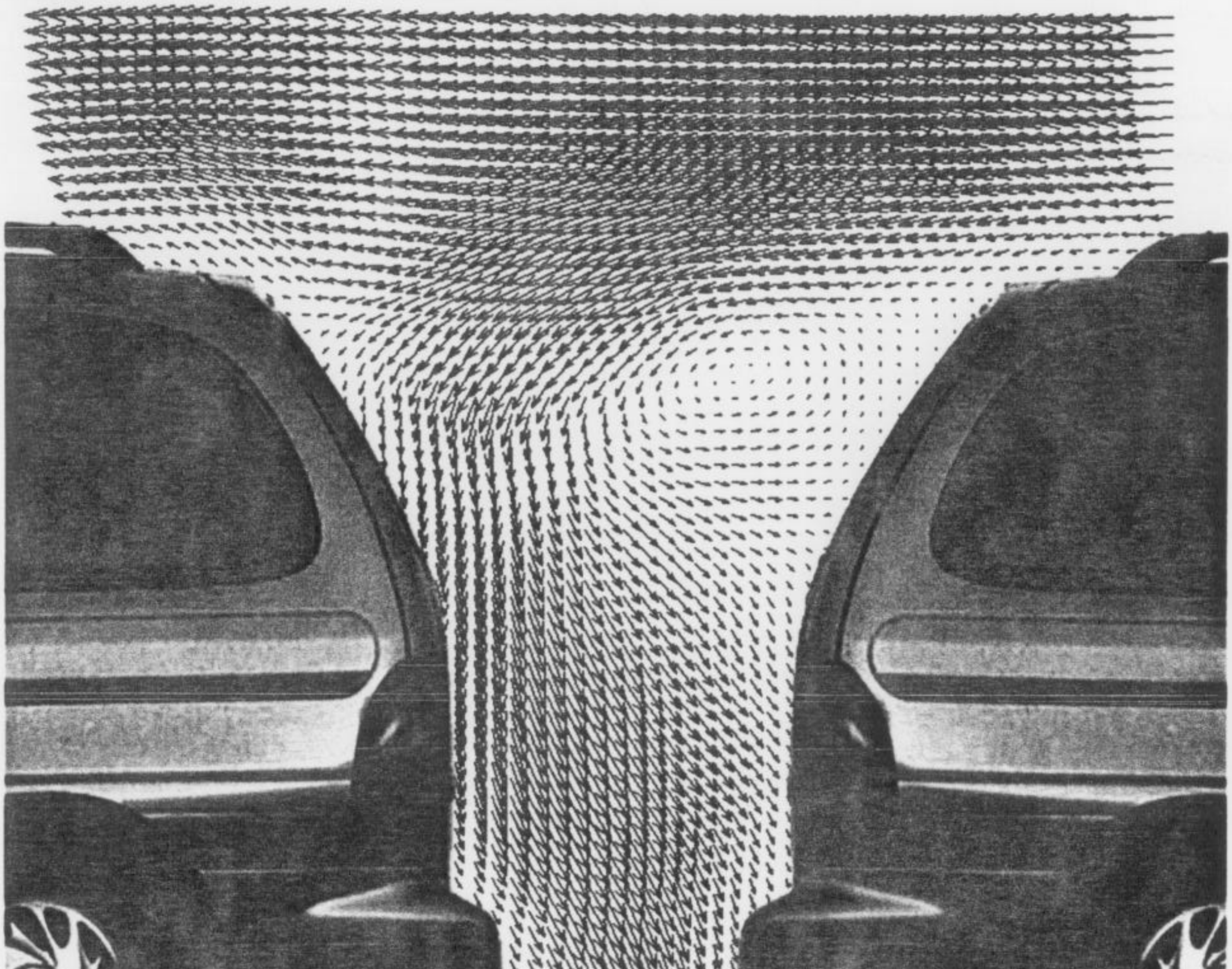
Duct attachment adapter
Flexible ducting
Gas propellant canisters (box of 10)
15 foot lead and remote control switch

CORONA INTEGRATED TECHNOLOGIES INC. **E.Mail: sales@smokemachines.com**
6215 Overstone Drive, West Vancouver, British Columbia, Canada, V7W 1X7. **Toll-free: 1-888-878-9433 Fax: 604-926-7422**

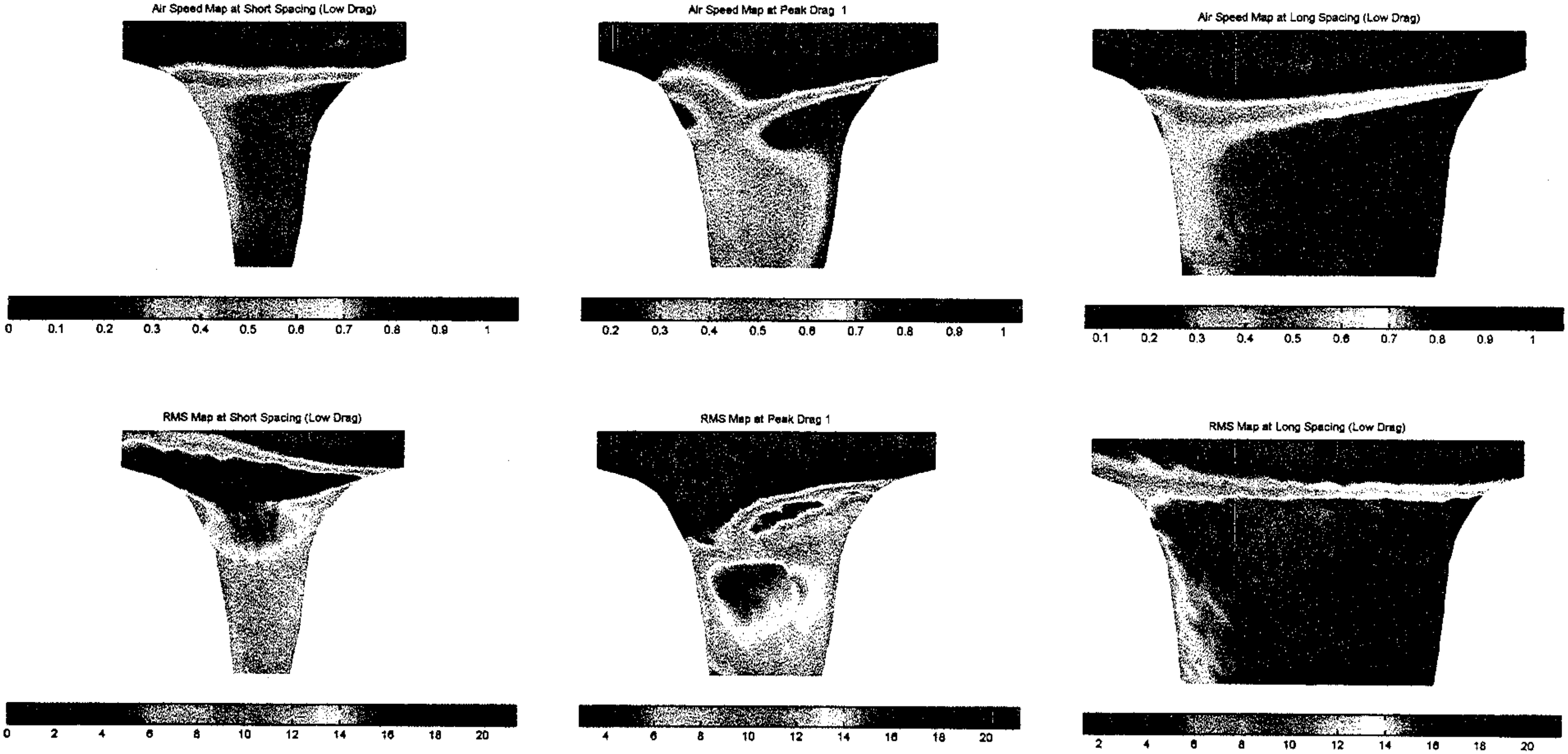

CP
0.726747
0.453495
0.180244
-0.0930083
-0.36626
-0.639512
-0.912764
-1.18602
-1.45927
-1.73252
-2.00577
-2.27902
-2.55227
-2.82553
-3.09878


CP
0.726747
0.453495
0.180244
-0.0930083
-0.36626
-0.639512
-0.912764
-1.18602
-1.45927
-1.73252
-2.00577
-2.27902
-2.55227
-2.82553
-3.09878

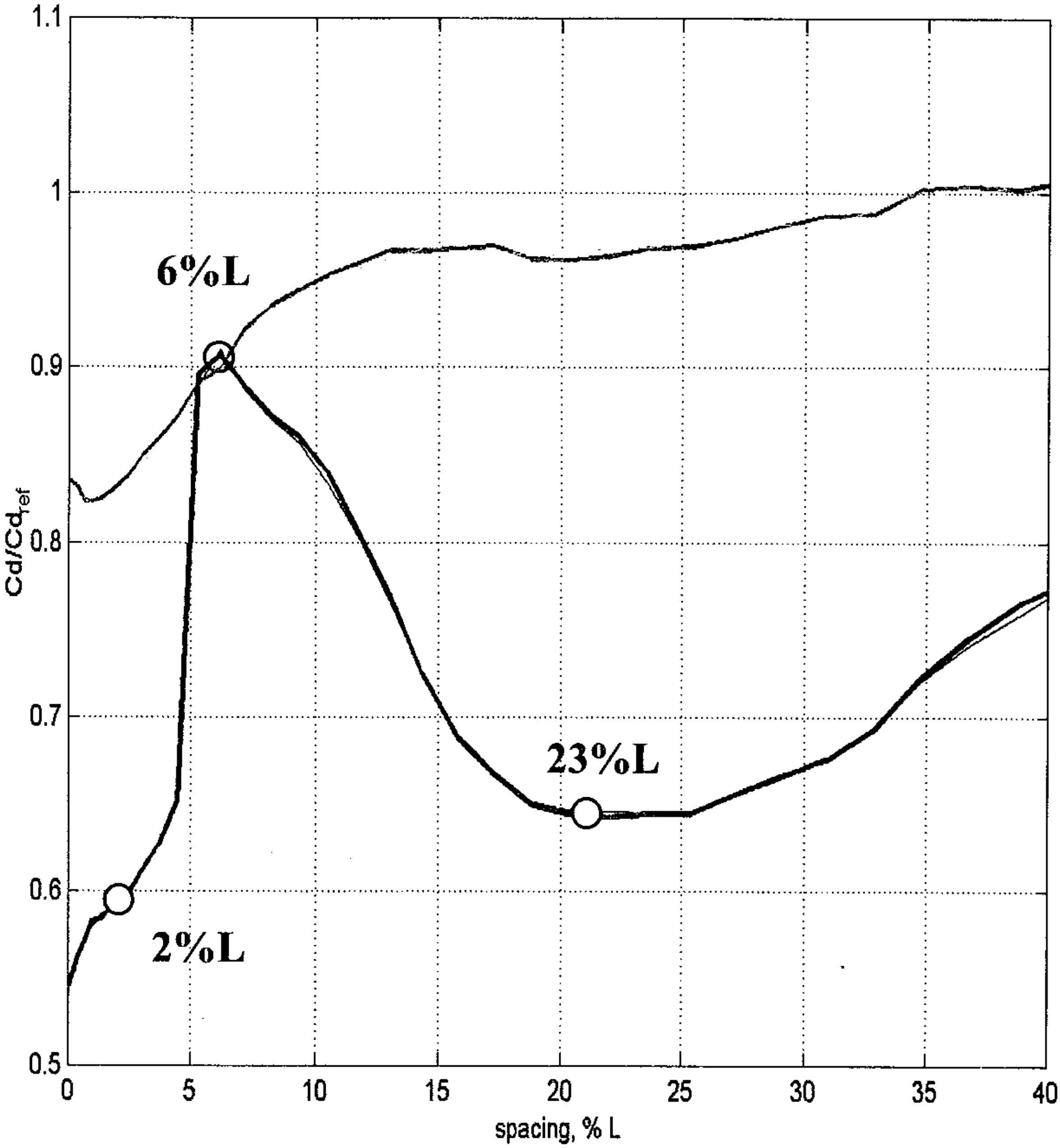




2%L 6%L 23%L



Air Speed Map at Short Spacing (Low Drag)
0 0.1 0.2 0.3 0.4 0.5 0.6 0.7 0.8 0.9 1
Air Speed Map at Peak Drag 1
0.2 0.3 0.4 0.5 0.6 0.7 0.8 0.9 1
Air Speed Map at Long Spacing (Low Drag)
0.1 0.2 0.3 0.4 0.5 0.6 0.7 0.8 0.9 1
RMS Map at Short Spacing (Low Drag)
0 2 4 6 8 10 12 14 16 18 20
RMS Map at Peak Drag 1
4 6 8 10 12 14 16 18 20
RMS Map at Long Spacing (Low Drag)
2 4 6 8 10 12 14 16 18 20



1.1
1
0.9
0.8
0.7
0.6
0.5
Cd/Cd$_{ref}$
6%L
23%L
2%L
0
5
10
15
20
25
30
35
40
spacing, % L

Steady RANS Computations

- Overset grid approach utilized a topology that
  - accurately represented the model to be tested in the 7x10 WT
  - resolved the major flow features
  - established a baseline grid size suggesting the grid sizes that might be required to refine the solution
- OVERFLOW solution
  - Converged to "a steady-state"
  - Needs to be compared to experimental data to determine if the averaged equations give reasonable numbers compared with "real" time averages for this grossly unsteady flow
  - Overall, there is no reason to expect useful steady results although from an engineering perspective the results may be close. The validation experiment will help determine the usefulness.

Karlin Roth, NASA Ames Research Center
kroth@mail.arc.nasa.gov
(650) 604-6678

NASA Ames Wind Tunnel Test Plans

- Points of Contact
  - Bruce Storms (650) 604-1356
  - Kevin James (650) 604-0178
- 7- by 10-Foot Wind Tunnel - Sandia Model
  - Target of opportunity for the Unified Instrumentation Test
  - Purpose is validation of RANS CFD capability for trucks
  - Model has arrived at NASA Ames, and test prep continues for 1/99
  - Detailed measurements include: pressure sensitive paint (PSP), oil-film interferometry, Doppler global velocimetry (DGV), video model deformation, particle imaging velocimetry (PIV), limited standard surface pressures, and forces

Karlin Roth, NASA Ames Research Center
kroth@mail.arc.nasa.gov
(650) 604-6678

NASA Ames Wind Tunnel Test Plans

- 12-Foot Pressure Wind Tunnel - Industry Model (1/8th scale)
  - Reynolds number sensitivity of various drag “deltas”
    - Mirrors
    - Trailer base-drag reduction device(s)
    - Tractor-trailer gap distance
    - Cooling air passages
    - Undercarriage drag
  - Collaboration with DOE, industry and university researchers
  - Measurements
    - Forces using 6K semispan balance (1200 lb axial force) - look into 2D load cells for low Reynolds numbers
    - Surface pressure distribution using PSI system and possibly PSP
    - Off-body flow using either DGV or PIV - laser delivery system and seeding are issues to be worked
  - Planned for FY00

Aerodynamic Design of Heavy Vehicles

Overview of the Computational Plans (RANS, LES)

Kambiz Salari

Aerosciences and Compressible Fluid Mechanics Dept. 9115

Sandia National Laboratories

August 1998

Sandia National Laboratories

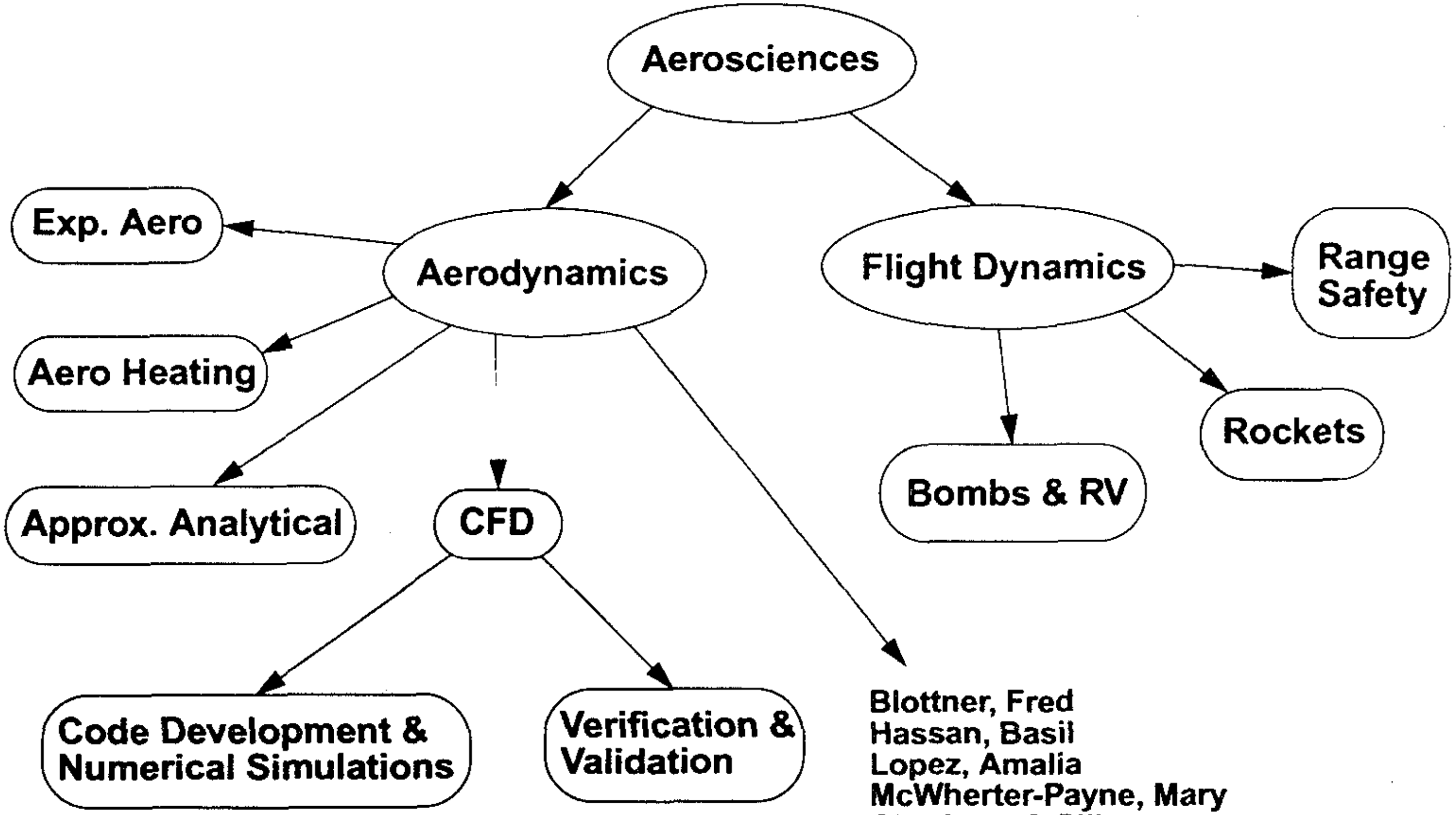
Outline of Presentation

- Aerosciences Department
- Computational Capabilities, RANS
- DOE Truck Project
- Plans for FY 99
- Philosophy of Validation Experiments

Engineering Sciences, Aerosciences Department

Sandia National Laboratories

Engineering Sciences Center



Blottner, Fred
Hassan, Basil
Lopez, Amalia
McWherter-Payne, Mary
Oberkampf, Bill
Payne, Jeffrey
Salari, Kambiz
Sterk, Todd

Aero Applications

- History
  - Primarily high speed flow simulations
  - Recently, there is an effort in low speed flow simulations
- Advanced Computational Capabilities
  - SACCARA (Sandia Advanced Code for Compressible Aerothermodynamics Research and Analysis)
  - CFD-ACE (Navier-Stokes code)
  - CHAD (Navier-Stokes code)
  - NS3D (Navier-Stokes code)
  - SPRINT (PNS code)
  - SANDIAC (Euler code)
  - MGAERO (Euler code)
  - HIBLARG (Boundary layer code)

Code Development & Numerical Simulation

- Physics Enhancement through Internal Research Programs
  - ESRF/LDRD
  - ESRF/Tech Base
- Range of Modeling and Simulation
  - Full Navier-Stokes code
    - Large Scale Computing (ASCI)
  - PNS codes
  - Euler Codes
  - Boundary Layer codes

SACCARA Current Capabilities:

- Based on parallel version of INCA™ Full Navier-Stokes code
- Implicit, Multi-block, structured grids for 2-D, Axisymmetric, and 3-D flows
- Finite volume discretization (steady and unsteady flows)
- Subsonic --> Hypersonic flow fields
- Ideal, equilibrium, and thermo-chemical nonequilibrium finite-rate gas chemistry
- Zero-,one-, and two-equation turbulence
- MP implementation on a variety of distributed parallel architectures (IBM, Intel, etc.)

Improving Physical Models in SACCARA

- Methods to model transition
  - Engineering models based on boundary layer
  - Parabolized Stability Equations (PSE) approach
- Turbulence models
  - One-equation Spalart Allmaras model
  - New two-equation k-ω model
  - New two-equation k-ζ model

DOE Truck Aero Project

- History
  - SNL GTS Work (RAMPANT), LDRD
  - Ahmed-body flow simulation (CHAD), USCAR/SCAAP
- Currently working on
  - Gridding the SNL GTS model
  - Running flow simulations for the GTS model with SACCARA

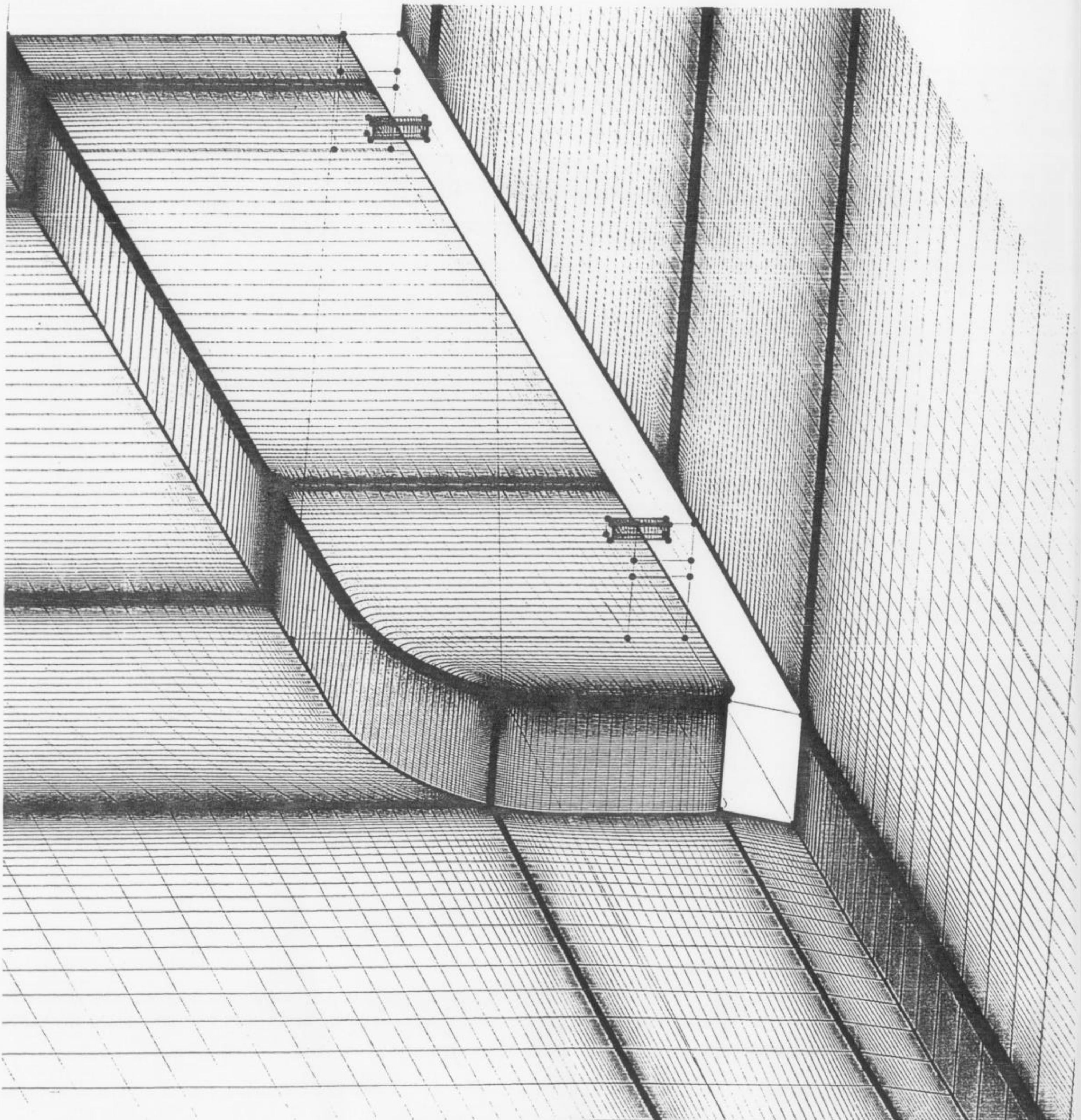
GTS Flow Simulation

Ground Transportation System (GTS) vehicle

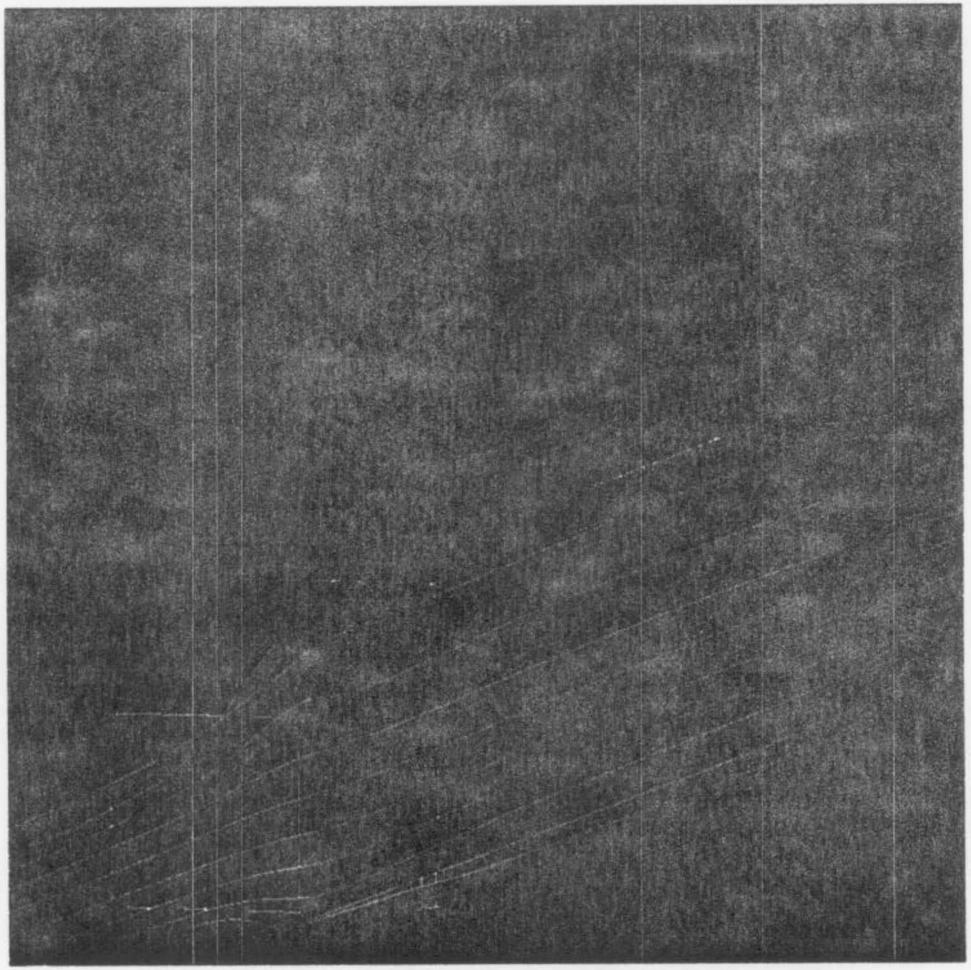
Texas A&M 7'x10' low speed tunnel test

Test condition:

- Run = 31, Re = 1.6×10^6, Wheels removed
- Yaw angle = 0 (deg.)
- Free stream velocity = 78 (m/s)
- Density = 1.17 (kg/m^3)
- Static pressure = 99,470.6 (Pa)
- Kinematic viscosity = 1.555×10^{-5} (m^2/s)



GTS Flow Simulation

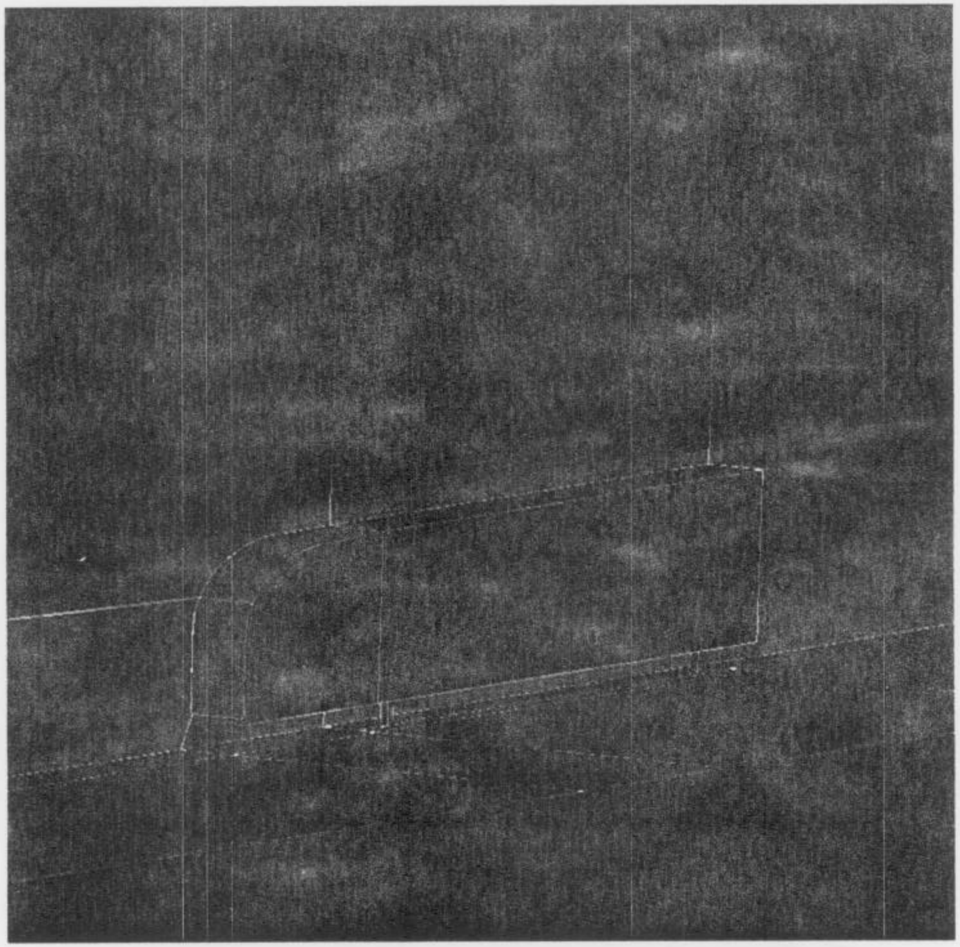


Particle traces, Re = 1.6×10^6

GTS Flow Simulation

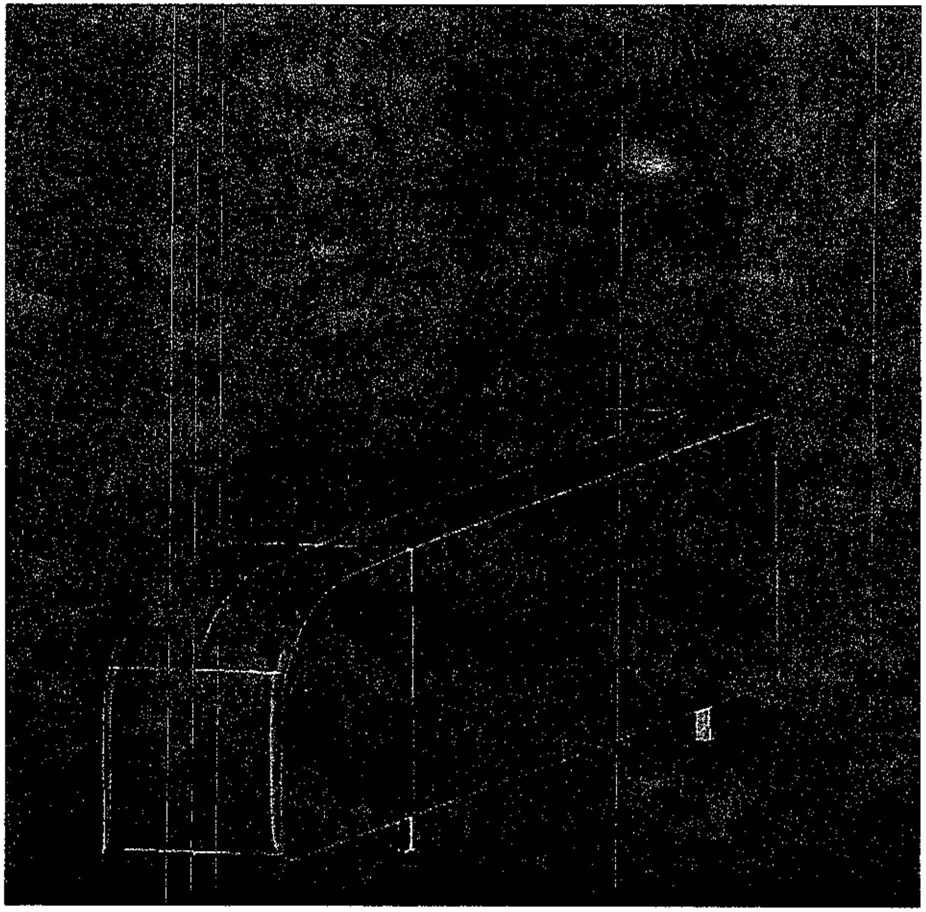
Sandia National Laboratories

Engineering Sciences Center



Contour of velocity magnitude, symmetry plane, Re = 1.6×10^6

GTS Flow Simulation



Pressure distribution on the surface, Re = 1.6×10^6

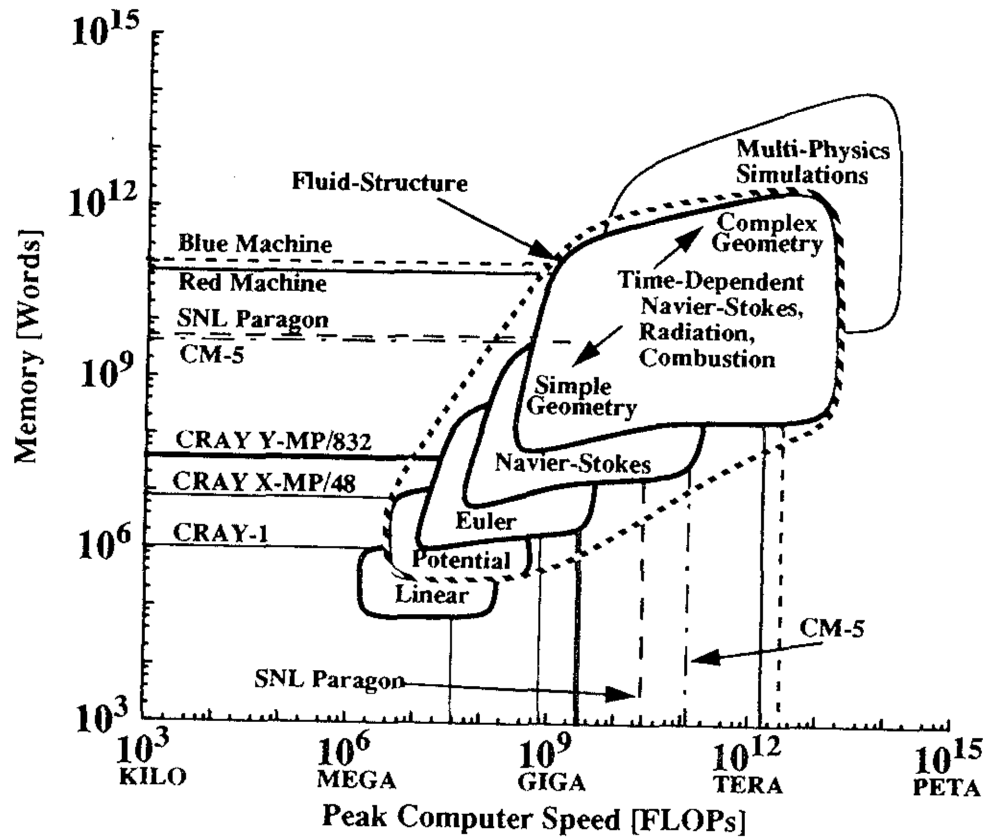
Plans for FY 99

- Continue to compare with SNL GTS shape
- Work with NASA 7'x10' test (Dec. 1998 ?)
- Work with other experimental programs (USC, 12' NASA/ARC test)
- Initiate gap/step study in conjunction with the rest of the project team
- Numerical Simulation Test cases
  - High Reynolds number RANS calculations
    - NASA 7'x10' test comparison for the Baseline (with gap/step if available)
  - Monitor Low-Reynolds tests (at USC)
- Add LES Capability to SACCARA

Leveraging from Other R&D Projects

- DOE ASCI Aero Program
  - Software models (algorithms, turbulence model)
  - Verification and Validation (V&V)
  - MP on Teraflops
- ASCI Level 1 Alliance with Stanford
- SNL ESRF Tech Base
  - Overset grid technology for MP computing
  - Improved physics models
- SNL CSRF LDRD
  - Mark Christon LES work
- DOE/TTI USCAR/SCCAP (FY 98)
  - Improved an unstructured flow solver CHAD
  - Ahmed-body flow simulation

Computational Fluid Dynamics is one of the “Grand Challenges” for the 1990’s



- The global nature of incompressible flow poses additional algorithmic and computational challenges
- Straightforward compressible flow time-marching algorithms are not applicable

"Big-Eddy" — Advanced Large Eddy Simulation Algorithms for Complex Flow Physics & Geometry

- The objective is to advance algorithms and methods for LES for unstructured grids, irregular geometry, and coupled physics
- A need to understand the interaction between:
  - Dispersive and diffusive errors
  - The influence of grid anisotropy
  - Filters and filter scales
  - Advective schemes and sub-grid scale models
- This effort seeks to advance LES models and methods by:
  - reducing the uncertainty and improving the reliability of large-eddy simulations
  - quantifying the effects of filters, filter scales, under-resolved flow fields, diffusive and dispersive errors, and stochastic SGS models

Philosophy of Code Validation Experiments

Sandia National Laboratories

Engineering Sciences Center

(1) A validation experiment should be jointly designed and executed by experimentalists and code builders

- Teamwork and candor are essential

(2) A validation experiment should be designed to capture the relevant physics, all initial and boundary conditions, and auxiliary data

- Leave no loop holes

(3) A validation experiment should utilize any inherent synergisms between experimental and computational approaches

- Offset strengths and weaknesses

Philosophy of Code Validation Experiments (cont.)

Sandia National Laboratories

(4) The flavor of a blind comparison of computational results with experimental data should be a goal

- It should be a "true prediction"

(5) A hierarchy of complexity of physics should be attacked in a series of validation experiments

- Identify levels of complexity and difficulty of prediction

(6) Develop and employ experimental uncertainty analysis procedures to delineate and quantify systematic and random sources of error

- Use symmetry arguments to help identify systematic errors

Truck Aerodynamics:

Large-Eddy Simulation (LES) using the Finite-Element Method (FEM)

Rose McCallen, Ph.D.
Lawrence Livermore National Laboratory

August 1998

University of California
Lawrence Livermore
National Laboratory

What do advanced tools provide and what are the challenges in developing and using these tools?

Background

- LES/FEM
- R&D issues

Approach and Deliverables

- Taking advantage of ASCI resources and past R&D
- SGS, wall modeling, boundary conditions
- Problem setup
- Data analysis

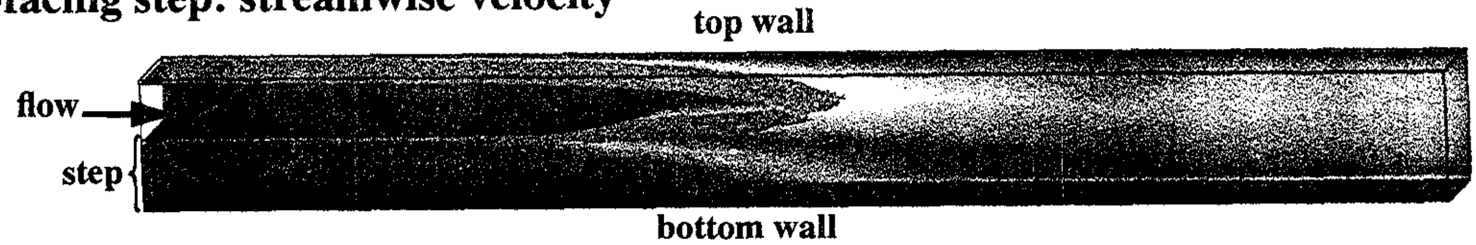
Status

The state-of-the-art CFD approaches provide inadequate information and accuracy.

Commercial state-of-the-art

- Reynolds-Averaged Navier-Stokes (RANS) turbulence model
- Many empirical parameters
- 2D, steady, time-averaged solution

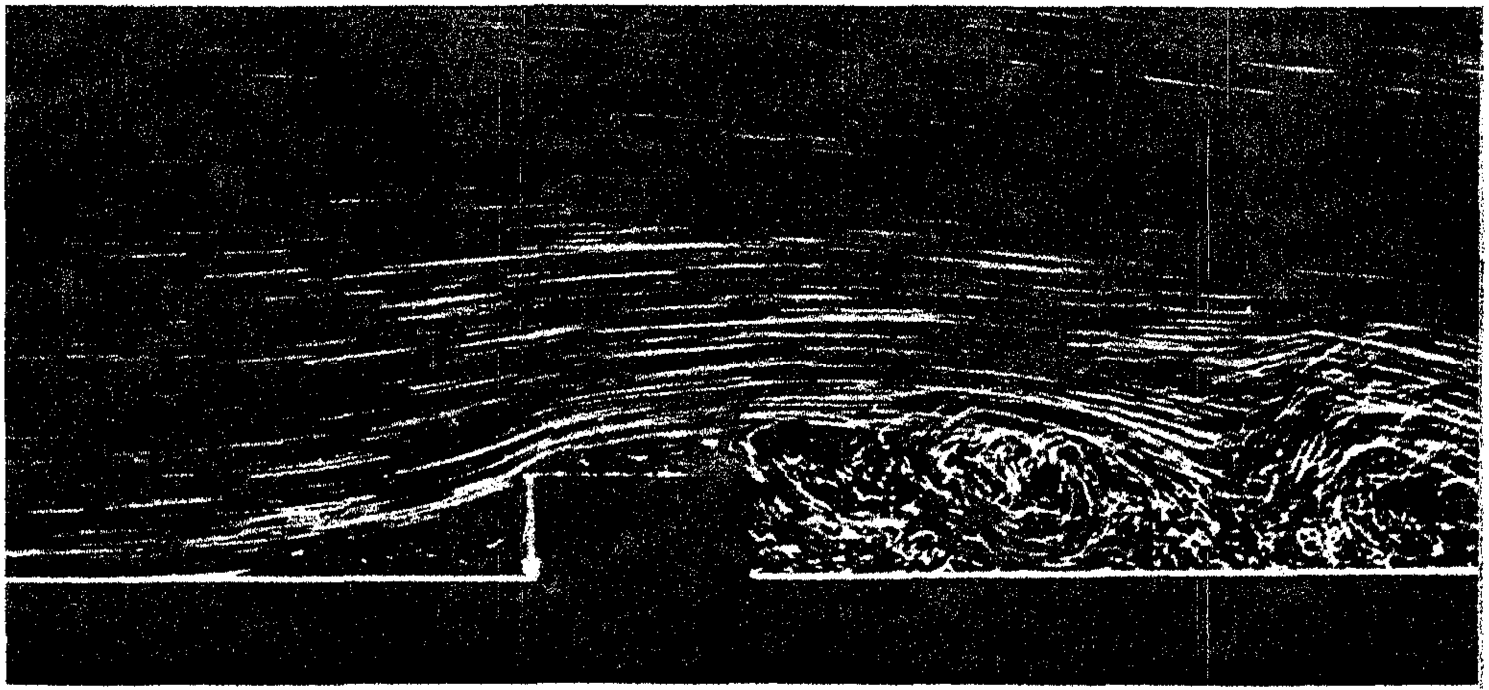
Backward-facing step: streamwise velocity



Current leading-edge technology

- Large-eddy simulation (LES) turbulence model
- One empirical parameter (maximum)
- 3D, unsteady solution of vortex shedding

Turbulent flow contains eddies ranging from large-scale to small-scale.



Ref. VanDyke, An Album of Fluid Motion

Large-eddy simulation <u>captures</u> the <u>large</u>-scale motion and <u>approximates</u> the <u>small</u>-scale motion.

all turbulent motions = large-scale motions + small-scale motions

= 'resolved' scale + 'subgrid' scale

$$u_\alpha = \bar{u}_\alpha + u'_\alpha$$

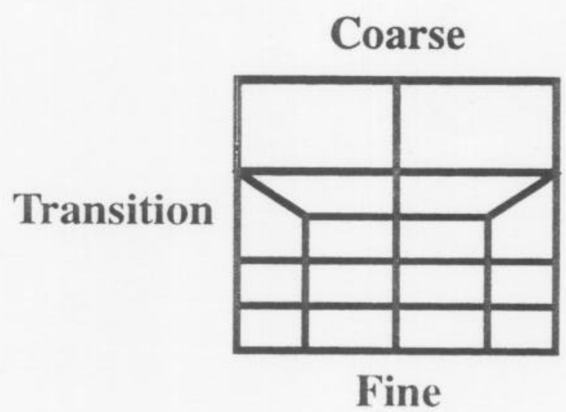
LES/FEM provides a unique approach for solving practical problems.

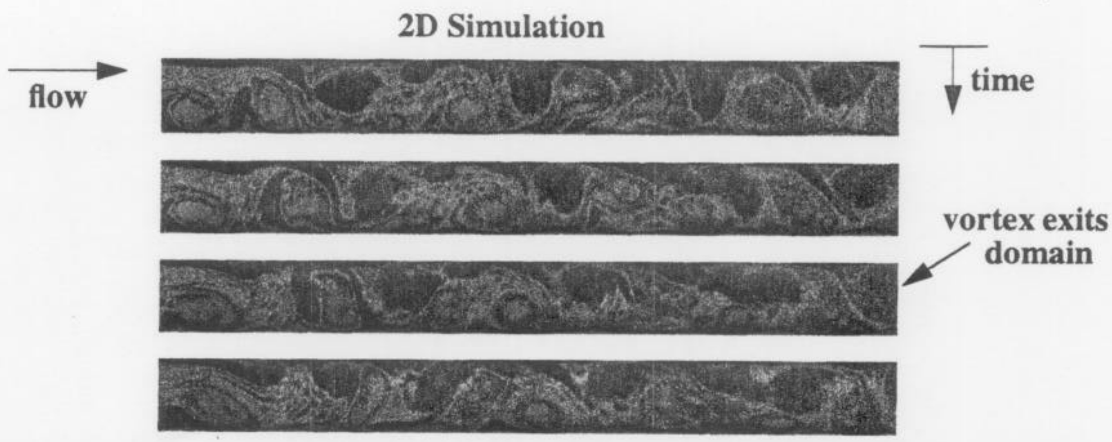
Advantages of LES

- Captures 3D time-dependent motion
- Less empiricism than other methods

Advantages of FEM

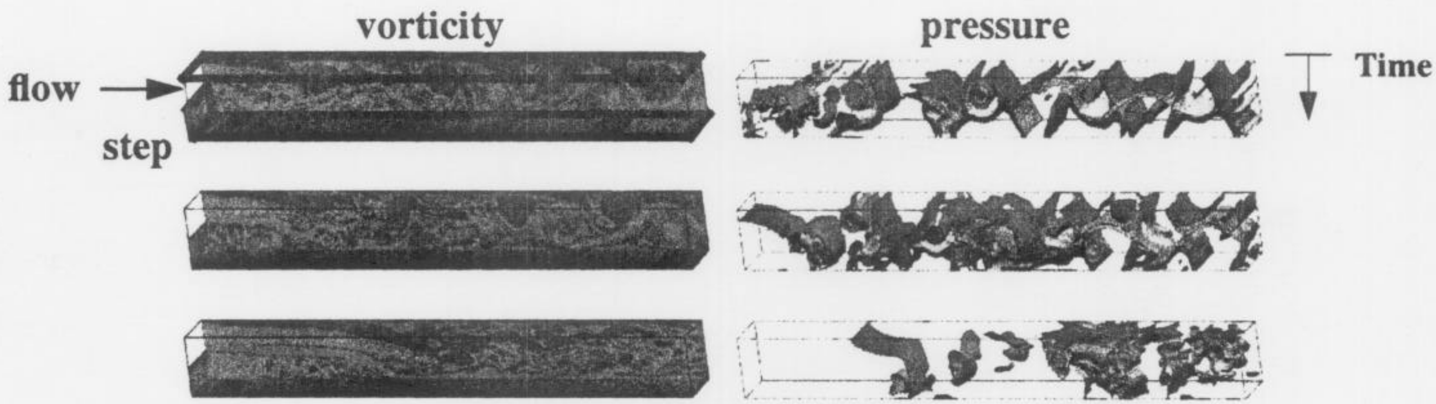
- Unstructured meshes
- Natural boundary conditions
- Coupling to other FEM codes





Zero natural boundary conditions capture the vortical outflow

The challenges are related to physical as well as computational modeling.



Boundary Conditions	**No slip/slip, outflow/inflow, periodic**
Size and Runtime	**Resolution of small eddy motion, evolution over long time scales**
Mesh Refinement	**Adaptive, unstructured**
Turbulence Models	**Approximations to reduce problem size and runtimes**
Analysis	**Large data sets, visualization, convergence testing**
Numerics	**Appropriate scheme, parallelization, solvers**

Our plan is to take advantage of existing methods and codes.

Integrate an incompressible flow model into an existing mulit-physics code

ALE3D (ASCI)
structural/thermal/chemistry/compressible-flow

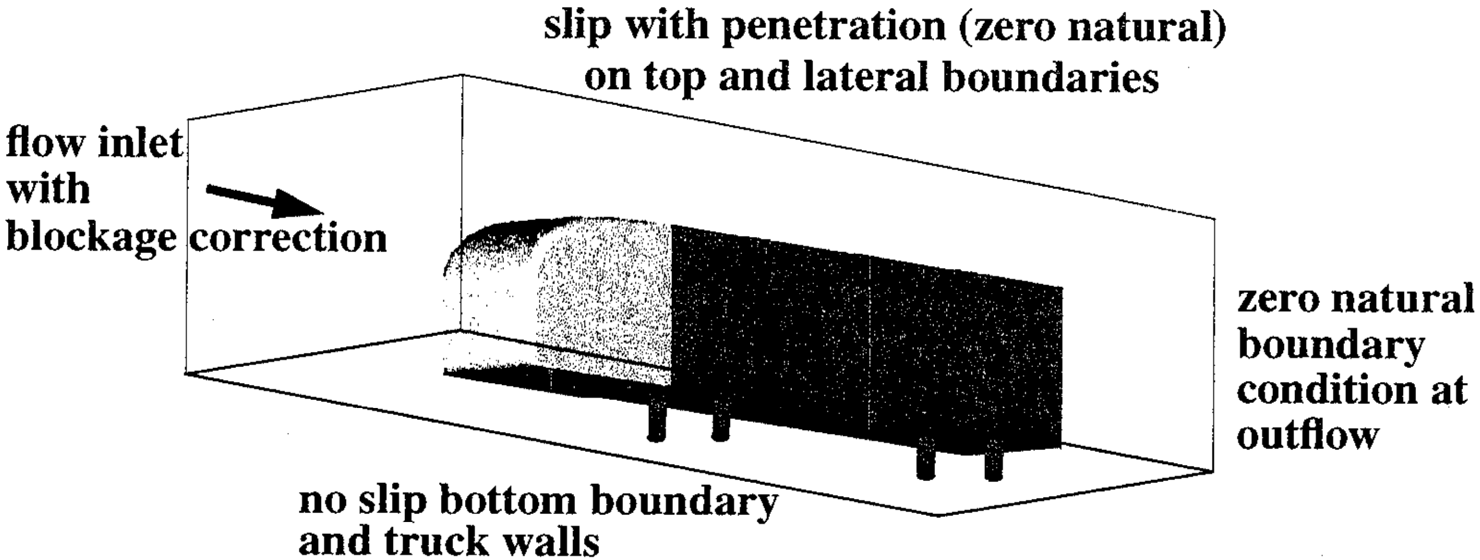
cook-off

Incompressible flow (Lab-Wide LDRD)
LES/FEM, data analysis methods, engineering application

The first year deliverable is to intergrate and develop the flow model and complete the demonstration problem.

Milestone	FY99 incompressible flow demonstration
R&D	Solver integration/parallelization
	Turbulence modeling
	Boundary conditions
	Data analysis

Computaional domain is chosen to minimize grid size



Fixed wall boundaries pose a challenge with LES.

A fine computational grid is required to capture near-wall flow

To simulate near wall effects, the eddy-viscosity should account for the SGS-stresses approaching zero at a wall

$$R_{\alpha\beta} = -2\upsilon_T S_{\alpha\beta} \qquad \text{where} \qquad S_{\alpha\beta} \equiv \frac{1}{2}\left(\frac{\partial \bar{u}_\alpha}{\partial x_\beta} + \frac{\partial \bar{u}_\beta}{\partial x_\alpha}\right)$$

Need $\upsilon_T \Rightarrow 0$ as approach wall

However,

$$\upsilon_T = (C\Delta)^2 (2 S_{\alpha\beta} S_{\alpha\beta})^{1/2}$$

State-of-the-art choices for wall region approximations

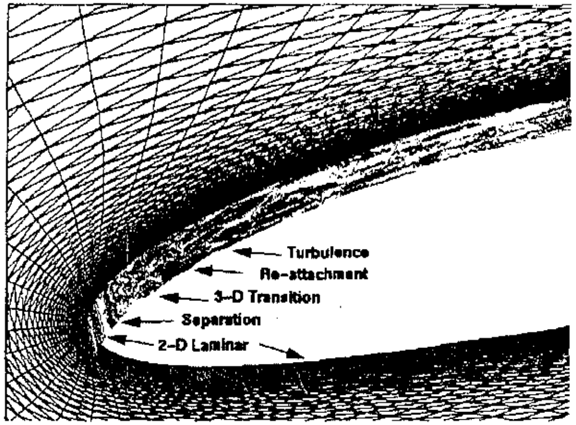
Dynamic SGS model

- increases the computational effort
- inherent instabilities

Reduce SGS based on distance from wall

- somewhat empirically based
- requires calculation of the normal distance to the wall

Unstructured Grid Large-Eddy Simulation of Flow Around a NACA 4412 Airfoil at 12°



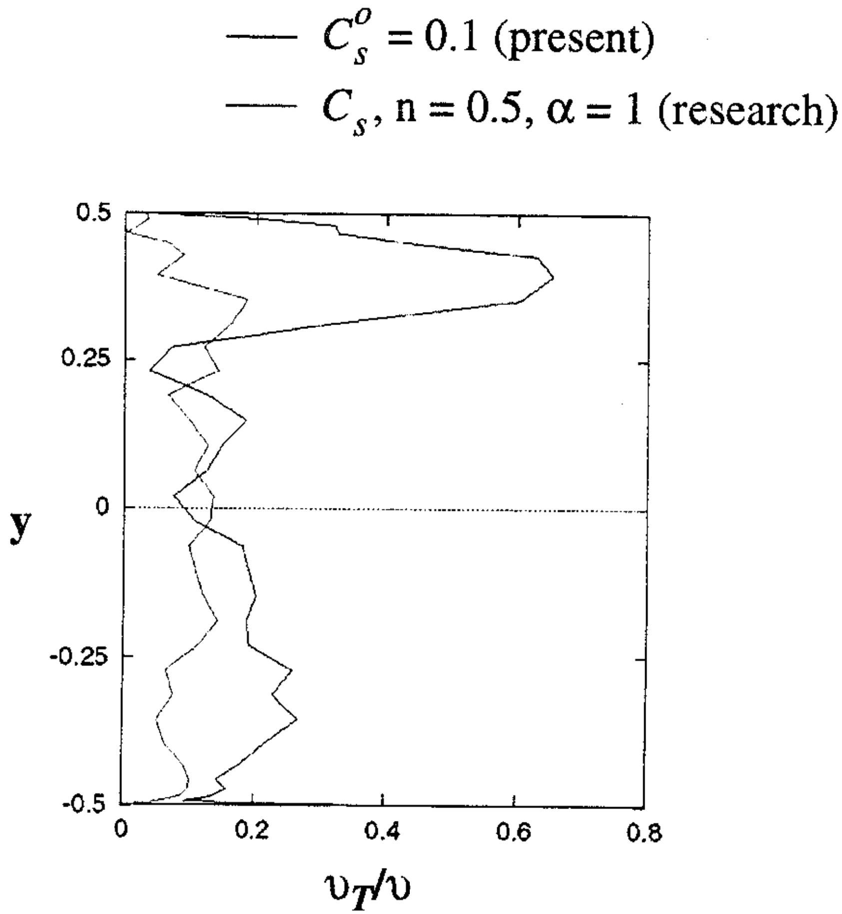
Reynolds number scaling is being investigated as a potential wall model. (UC Davis collaboration)

$$\upsilon_T = (C_s\Delta)^2[S_{\alpha\beta}S_{\alpha\beta}]^{\frac{1}{2}}$$

$$C_s \equiv C_s^o\delta$$

$$\delta = \begin{cases} \left(\dfrac{Re}{Re^o}\right)^n & for\ \ 0 \le Re \le Re^o \\ 1 & for\ \ Re > Re^o \end{cases}$$

$$Re^o \equiv \alpha Re_{max}$$

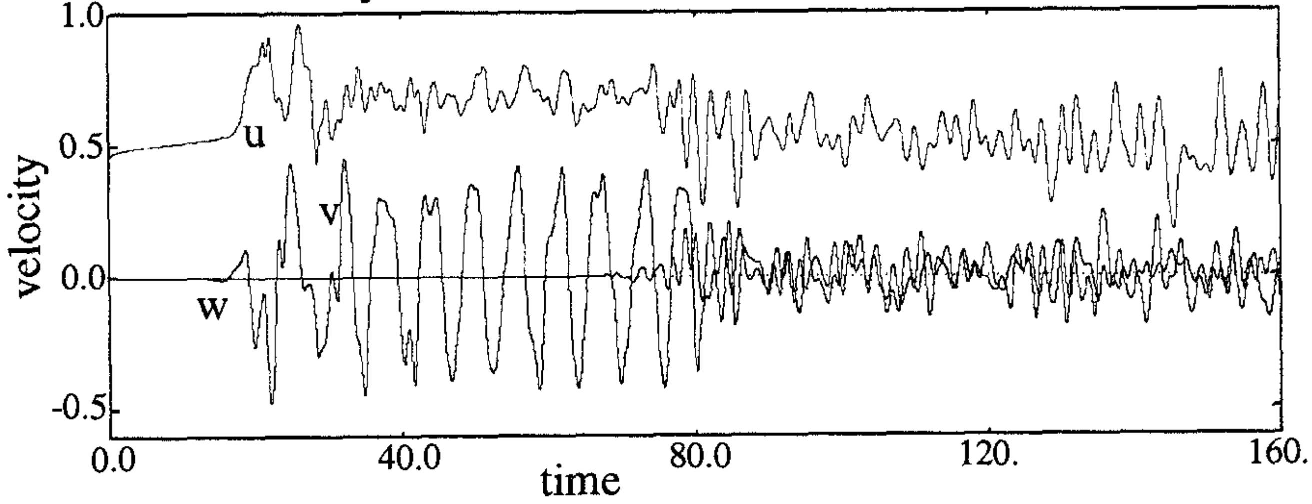


Advantages

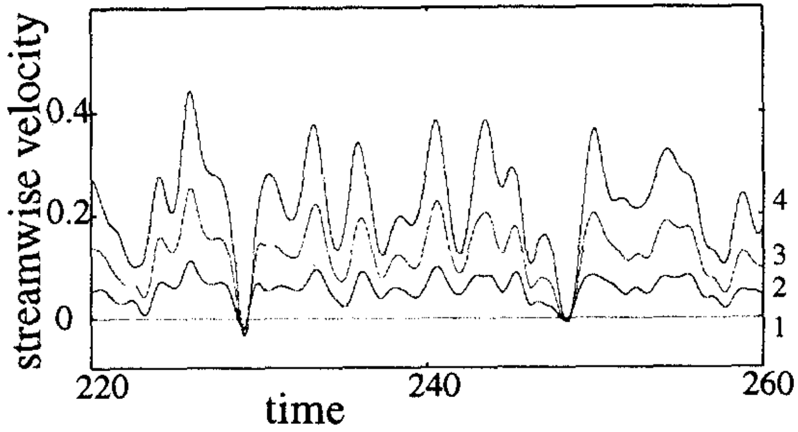
- **Corrects subgrid-scale model in wall regions without affecting core flow**
- **Computationally inexpensive (adds < 2%)**
- **Applicable to unstructured grids (normal to wall is not needed)**

Time histories provide local flow information

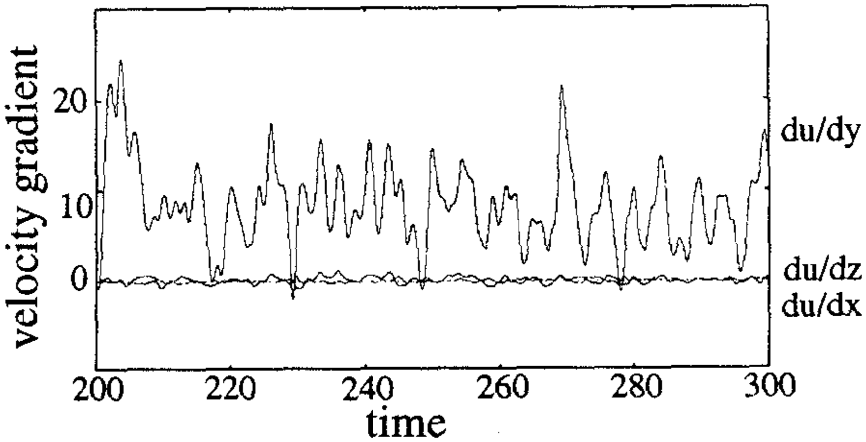
Velocity in core of flow - transition from 2-D to 3-D



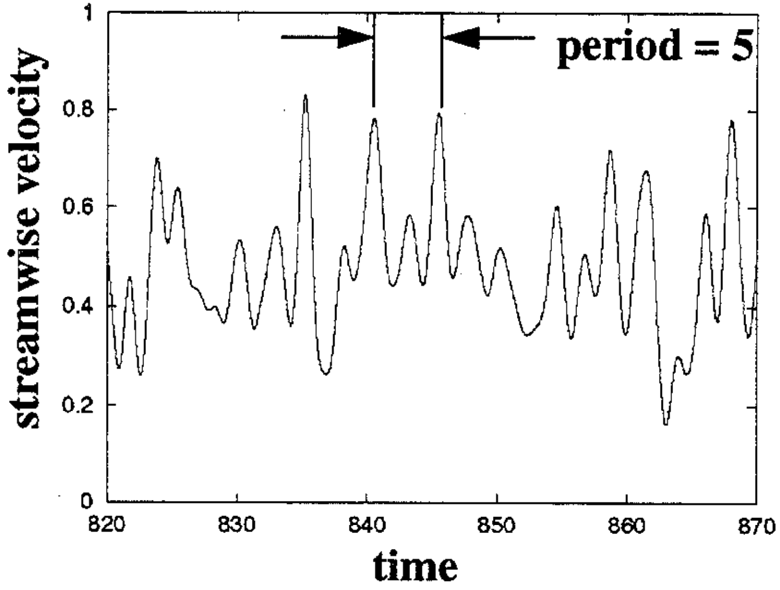
Velocity at points near wall

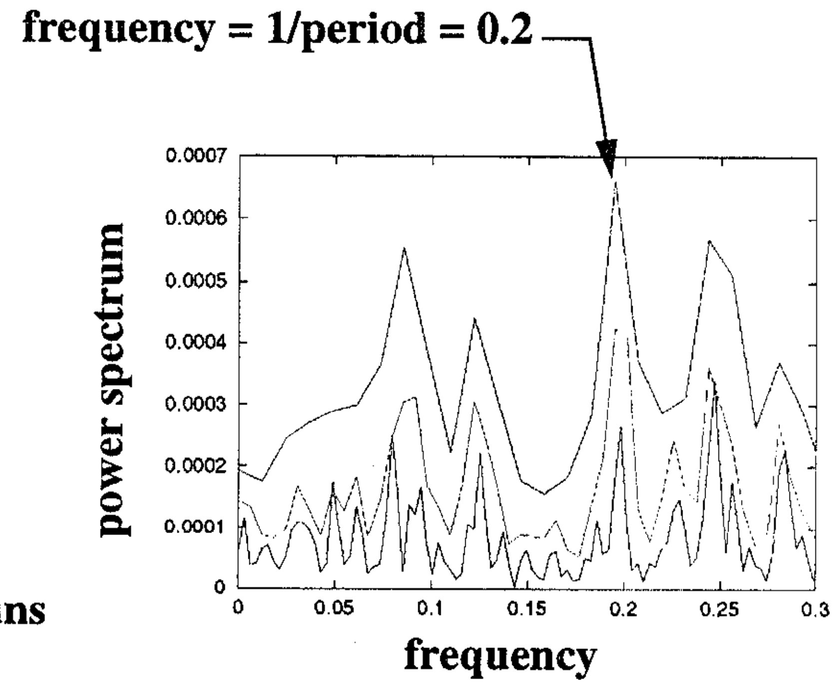


Comparison of gradients near wall



Power spectrum analysis can be used to determine the dominant frequencies.



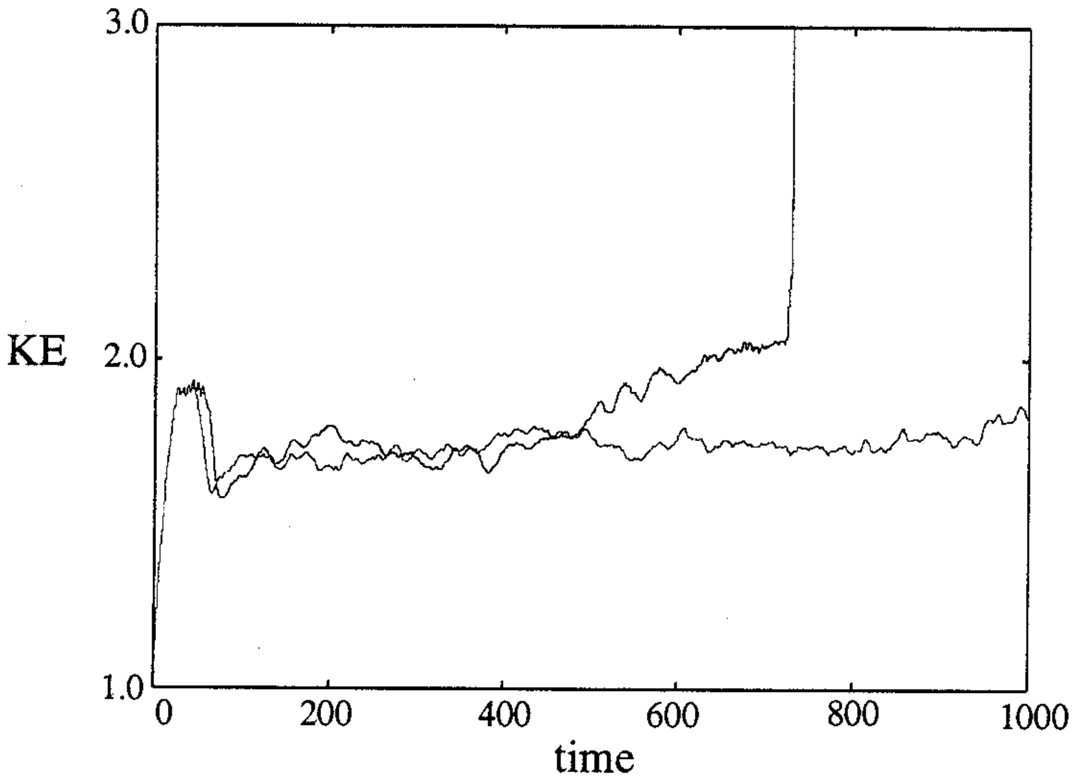


Goal: Use power spectrum to compare runs
- different meshes
- different time steps

Issue: Peak identification

We monitor global kinetic energy to insure stability.

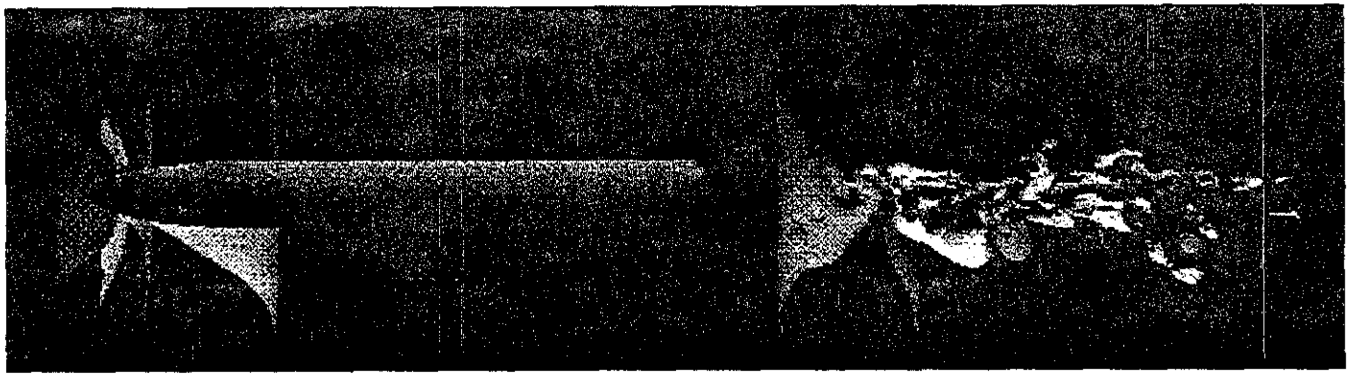
$$kinetic\ \ energy = KE(t, \Delta t, \Delta \underline{x}) = \frac{1}{2}\iiint_{\Omega} \underline{u} \cdot \underline{u} d\Omega$$



A variable time-step is beneficial.

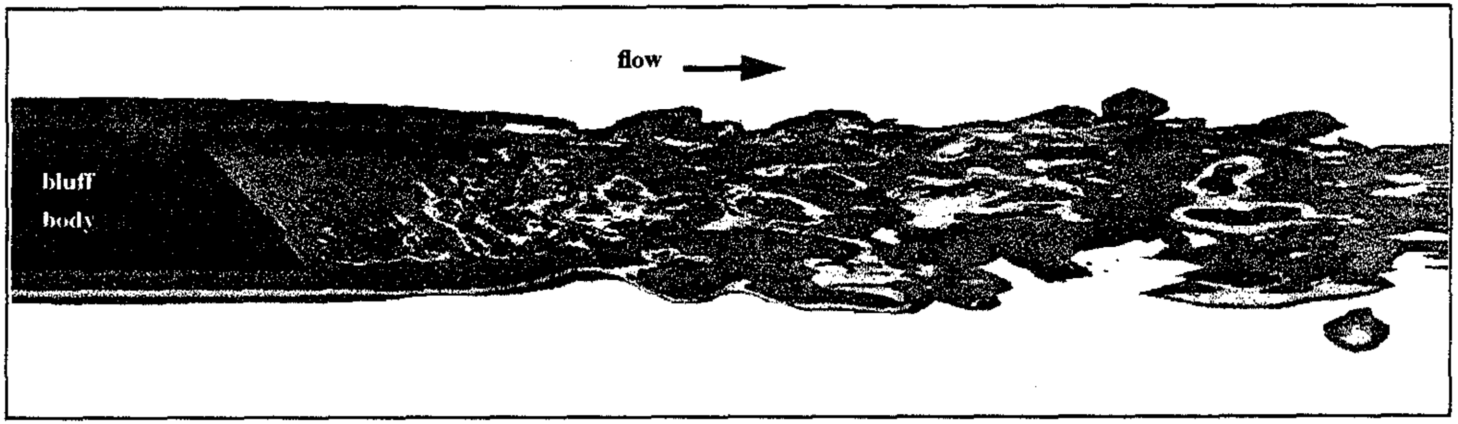
Flow visualization requires choosing the right parameters and movie making.

Pressure



Large-Eddy Simulation of vortex shedding in the wake of a bluff body.

Enstrophy



Status : Work has begun on code developement and problem setup.

Code development

Developed plan for integration of ALE3D and incompressible flow model

Seeking LDRD/Program support

Computational model

Problem definition

Grid generation from PROE file

LES is a challenge but we have the experience and resources to succeed.

State-of-the-art in CFD

- Inadequate for modeling truck aerodynamics

LES/FEM has advantages

- Less empiricism
- Built-in outflow conditions

Approach

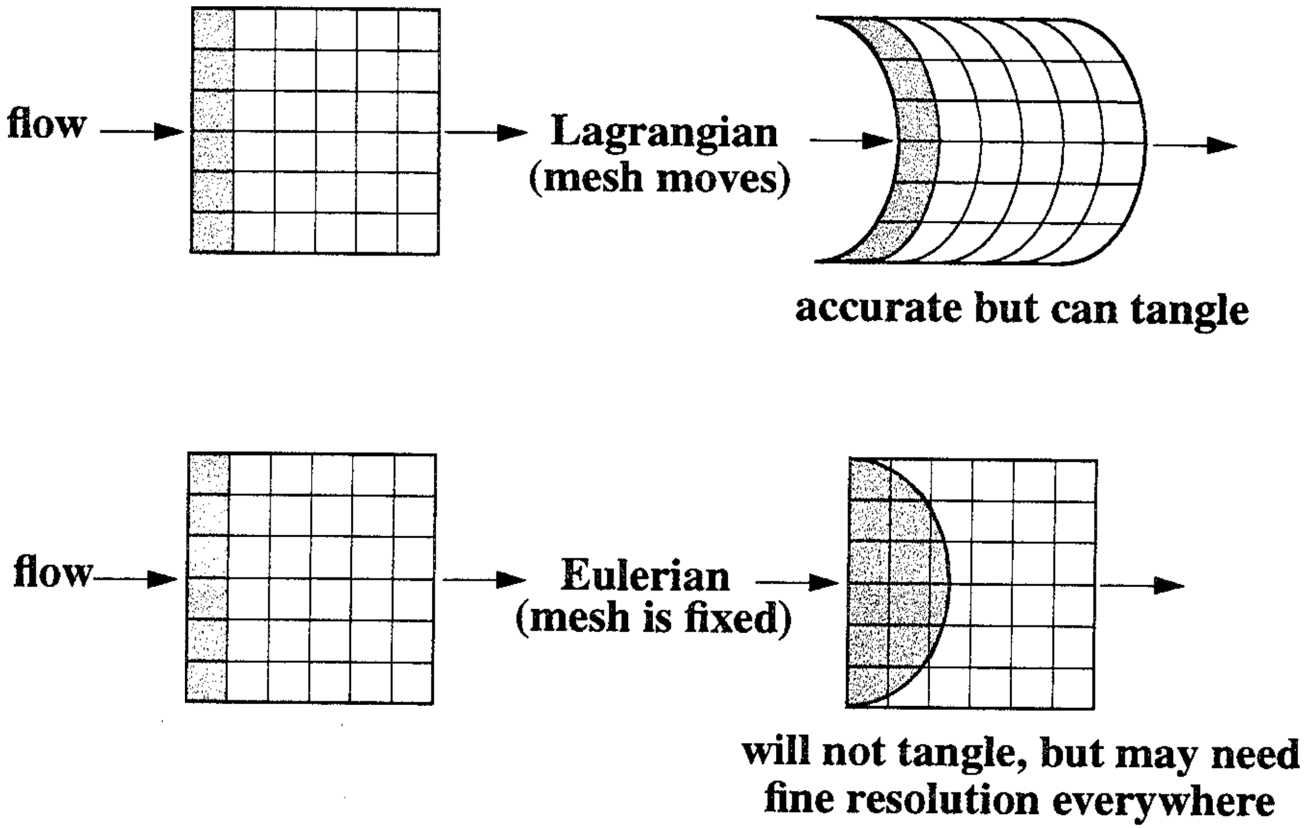
- Take advantage of existing methods and codes
- Keep it simple - Smagorinsky SGS model and reduced computational domain

Data analysis

- Time-averaging, visualization, time histories, and power spectrum

ALE3D combines the strengths of Lagrangian and Eulerian methods.

ALE3D: Arbitrary Lagrangian Eulerian Three Dimensional



ALE

Solution is Lagrangian or Eulerian or both

Mesh can 'relax' as needed

The LES/FEM formulation has advantages.

FEM

Mass

$$\left(\int_\Omega \psi_i \frac{\partial \phi_j}{\partial x_\alpha}\right)\bar{u}_\alpha{}^j = 0$$

Momentum

$$\left(\int_\Omega \phi_i \phi_j\right)\frac{\partial \bar{u}_\alpha{}^j}{\partial t} + \left(\bar{u}_\beta^k \int_\Omega \phi_i \phi_k \frac{\partial \phi_j}{\partial x_\beta}\right)\bar{u}_\alpha{}^j + \left(\int_\Omega \upsilon \frac{\partial \phi_i}{\partial x_\beta}\frac{\partial \phi_j}{\partial x_\beta}\right)\bar{u}_\alpha{}^j$$

$$-\left(\int_\Omega \psi_j \frac{\partial \phi_i}{\partial x_\alpha}\right)\bar{P}^{\,j} - \left(\int_\Omega R_{\alpha\beta}\frac{\partial \phi_i}{\partial x_\beta}\right)\bar{u}_\alpha{}^j = \int_{\partial\Omega} \phi_i \, f_\alpha$$

Boundary Condition $f_\alpha = n_\beta \bar{\tau}_{\alpha\beta} = n_\beta\left(-\bar{P}\delta_{\alpha\beta} + \upsilon\frac{\partial \bar{u}_\alpha}{\partial x_\beta} - R_{\alpha\beta}\right)$

The outflow boundary conditions are built into the FEM formulation.

LES

Subgrid-scale model $R_{\alpha\beta} = -2\upsilon_T S_{\alpha\beta}$, $S_{\alpha\beta} \equiv \frac{1}{2}\left(\frac{\partial \bar{u}_\alpha}{\partial x_\beta} + \frac{\partial \bar{u}_\beta}{\partial x_\alpha}\right)$

Eddy-viscosity $\upsilon_T = (C\Delta)^2(2S_{\alpha\beta}S_{\alpha\beta})^{1/2}$

We want to minimize the computational effort for *practical* applications.

FEM

Galerkin finite-element method

Discrete pressure Poisson equation

Velocities calculated with explicit forward Euler

Simplifications:

Tri-linear velocity and piecewise constant pressure basis functions

One-point Gaussian quadrature

Lumped mass matrix

Centroid advection velocity

LES

Smagorinsky subgrid-scale model

Approximated advection, $\overline{\bar{u}_\alpha \bar{u}_\beta} = \bar{u}_\alpha \bar{u}_\beta$

Neglected cross-terms, $\overline{u_\alpha' \bar{u}_\beta} + \overline{\bar{u}_\alpha u_\beta'} = 0$

The LES advection term doesn't have to be approximated.

Filter Navier-Stokes Equations

$$\frac{\partial \bar{u}_\alpha}{\partial x_\alpha} = 0$$

$$\frac{\partial \bar{u}_\alpha}{\partial t} + \overline{\bar{u}_\beta \frac{\partial \bar{u}_\alpha}{\partial x_\beta}} = -\frac{1}{\rho}\frac{\partial \bar{p}}{\partial x_\beta} + \upsilon \frac{\partial^2 \bar{u}_\alpha}{\partial x_\beta^2} - \frac{\partial}{\partial x_\beta}(\overline{u'_\alpha \bar{u}_\beta} + \overline{\bar{u}_\alpha u'_\beta} + \overline{u'_\alpha u'_\beta})$$

manipulate $\bar{u}_\beta \frac{\partial \bar{u}_\alpha}{\partial x_\beta}$ $\quad -\frac{\partial}{\partial x_\beta}(\overline{u'_\alpha \bar{u}_\beta} + \overline{\bar{u}_\alpha u'_\beta} + \overline{u'_\alpha u'_\beta} + \overline{\bar{u}_\alpha \bar{u}_\beta} - \bar{u}_\alpha \bar{u}_\beta)$

approximate $\quad -\frac{\partial}{\partial x_\beta}(\overline{u'_\alpha u'_\beta})$

FEM allows for the 'exact' solution of the LES advection term.

FEM

Mass $$\left(\int_\Omega \psi_i \frac{\partial \phi_j}{\partial x_\alpha}\right)\bar{u}_\alpha^{\,j} = 0$$

Momentum $$\left(\int_\Omega \phi_i \phi_j\right)\frac{\partial \bar{u}_\alpha^{\,j}}{\partial t} + \left(\bar{u}_\beta^k \int_\Omega \phi_i \frac{\partial}{\partial x_\beta}\overline{\phi_j \phi_k}\right)\bar{u}_\alpha^{\,j} + \left(\int_\Omega \upsilon \frac{\partial \phi_i}{\partial x_\beta}\frac{\partial \phi_j}{\partial x_\beta}\right)\bar{u}_\alpha^{\,j}$$

$$-\left(\int_\Omega \psi_j \frac{\partial \phi_i}{\partial x_\alpha}\right)\bar{P}^{\,j} - \left(\int_\Omega (\overline{u'_\alpha \bar{u}_\beta} + \overline{\bar{u}_\alpha u'_\beta} + \overline{u'_\alpha u'_\beta})\frac{\partial \phi_i}{\partial x_\beta}\right)\bar{u}_\alpha^{\,j} = \int_{\partial\Omega} \phi_i\, f_\alpha$$

Boundary Condition $$f_\alpha = n_\beta\left(-\bar{P}\delta_{\alpha\beta} + \upsilon\frac{\partial \bar{u}_\alpha}{\partial x_\beta} - (\overline{u'_\alpha \bar{u}_\beta} + \overline{\bar{u}_\alpha u'_\beta} + \overline{u'_\alpha u'_\beta})\right)$$

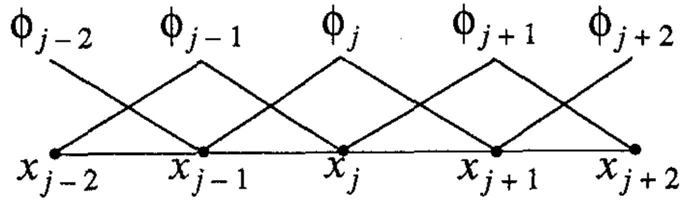
Expansions $$\bar{u}_\alpha^{\,h} = \sum_{j=1}^{N} \bar{u}_\alpha^{\,j}(t)\phi_j(\underline{x})$$

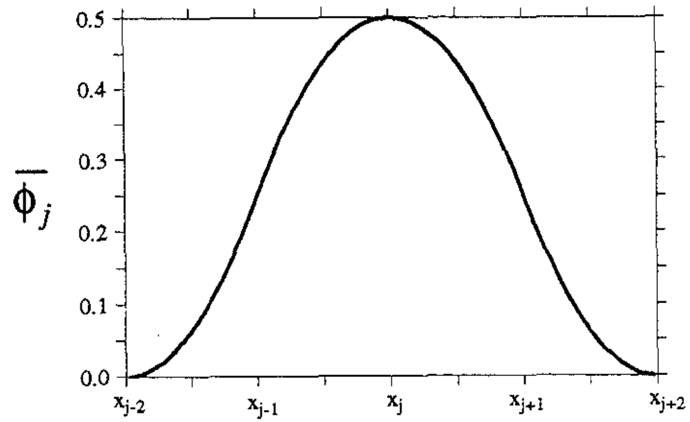
$$\bar{P}^h = \sum_{j=1}^{M} \bar{P}^j(t)\psi_j(\underline{x})$$

Variables are defined *continuously* at all points in the flow field.

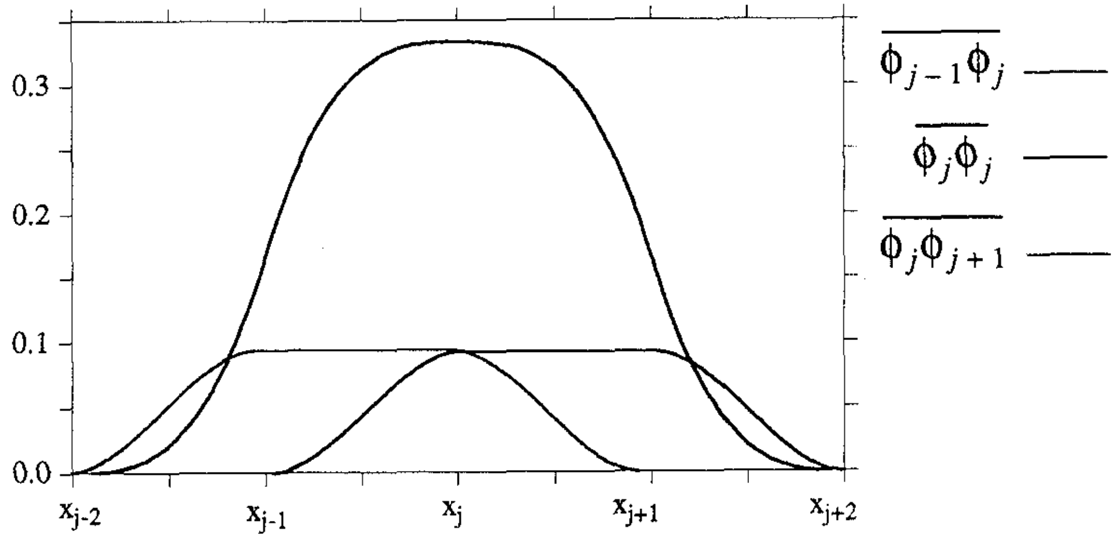
In 1-D, filtering $\phi_j\phi_k$ over two-grid lengths results in functions that span 3 to 4 grid lengths.

For linear basis functions, $\Delta_f = 2h$, and cell volume averaging:





Challenge:
With FEM, integrations are done at the element level, but $\overline{\phi_j\phi_k}$ for $\Delta_f > \Delta$ requires integration over multiple elements.



Vortex Methods for Flow Simulation

A. Leonard
California Institute of Technology

Numerical technique to solve the Navier-Stokes Equations

Suitable for Direct Simulation and Large-Eddy Simulation

Uses vorticity (curl of the velocity) as a variable

Computational elements move with the fluid velocity

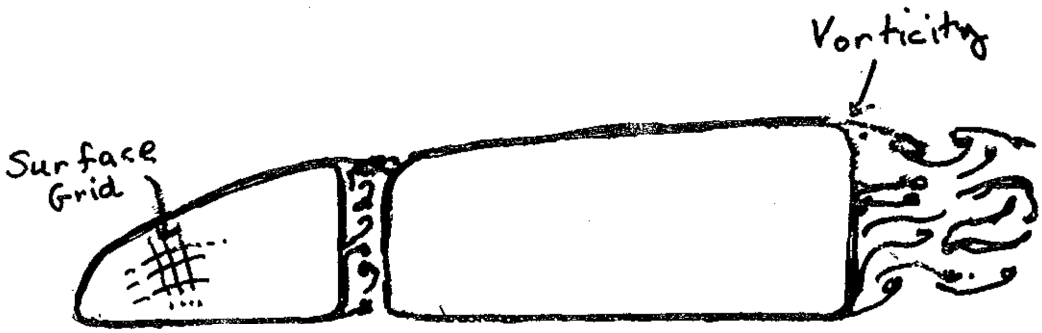
Advantages

Computational elements only where vorticity is nonzero

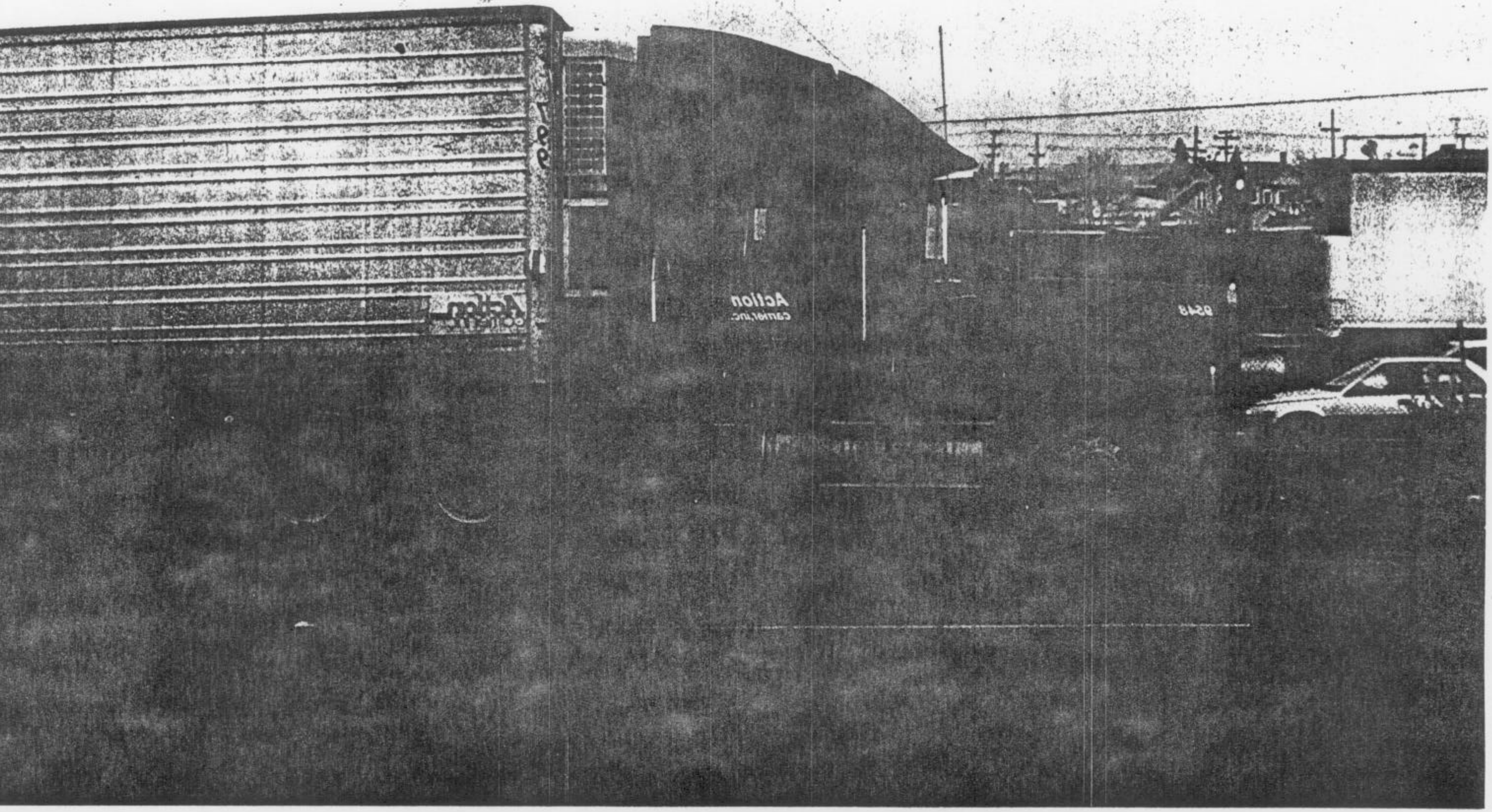
No grid in the flowfield

Only 2D grid on vehicle surface

Boundary conditions in the far field automatically satisfied



$Re \approx 5 \times 10^6$



Status / Future Work

Direct Simulation possible for Reynolds No. = 10^3 to 10^4
(Truck Speed, 0.01 mph)

N=10^{14} elements required for Reynolds No. = 20 x 10^6
(Truck Speed, 70 mph)

Must use Large-Eddy Simulation in the forseeable future

Treatment of small-scale (subgrid-scale) turbulence in the wake

Treatment of small-scale turbulence in the boundary layers *

Treatment of fluidic actuators, blowing/suction ,
vortex generators and other flow control devices

Implementation of Vortex Method for complex geometries *

Turbulent Boundary Layer (Adrian et al)

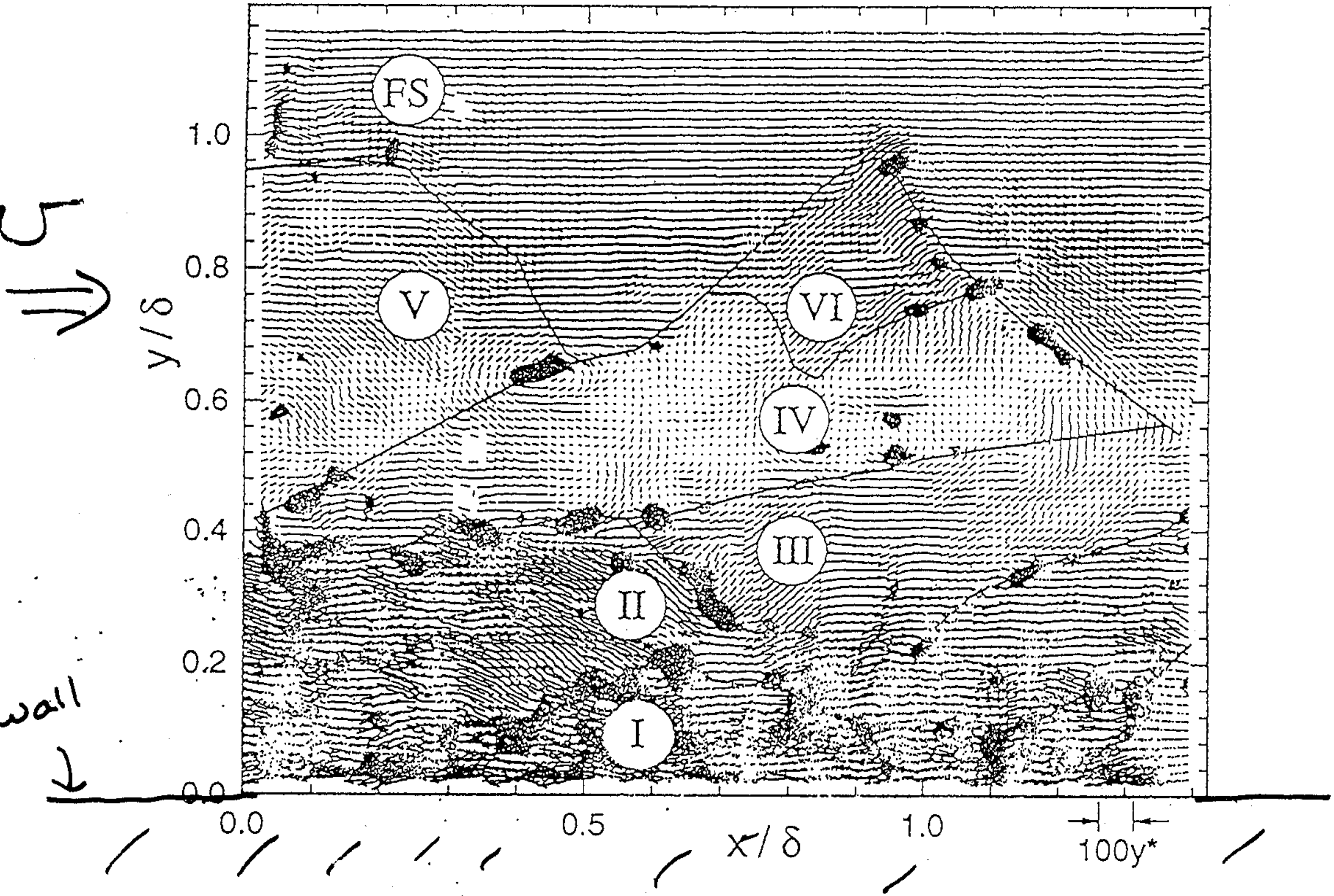


FIG. 1. Instantaneous velocity field in the streamwise-wall-normal plane of an $Re_\theta=6845$ boundary layer viewed in a frame convecting at $0.9U_\infty$. The black lines indicate the approximate boundaries of zones in which the streamwise momentum is nearly constant. The dark-gray shaded areas denote regions where spanwise vorticity, nondimensionalized by the friction velocity u_τ and the viscous wall length scale $y^* = \nu/u_\tau$, is less than -0.03.

TREATMENT OF SURFACE VORTICITY

Standard Panel Method for N Panels

Low order accuracy - First order accurate

Computationally and storage limited - $O(N^2)$ matrix elements computed and stored and $O(N^2)$ operations per solution

Only N= 10,000 to 20,000 feasible

Advanced Panel Method (Brady, Pullin, AL)

High accuracy - Third order accurate

Computationally efficient - $O(N)$ storage locations $O(N^{3/2})$ operations per solution [can go to $O(N^{4/3})$, $O(N \log N)$, $O(N)$]

N= 100,000 to 200,000 is no problem

Triangular mesh with automatic mesh refinement

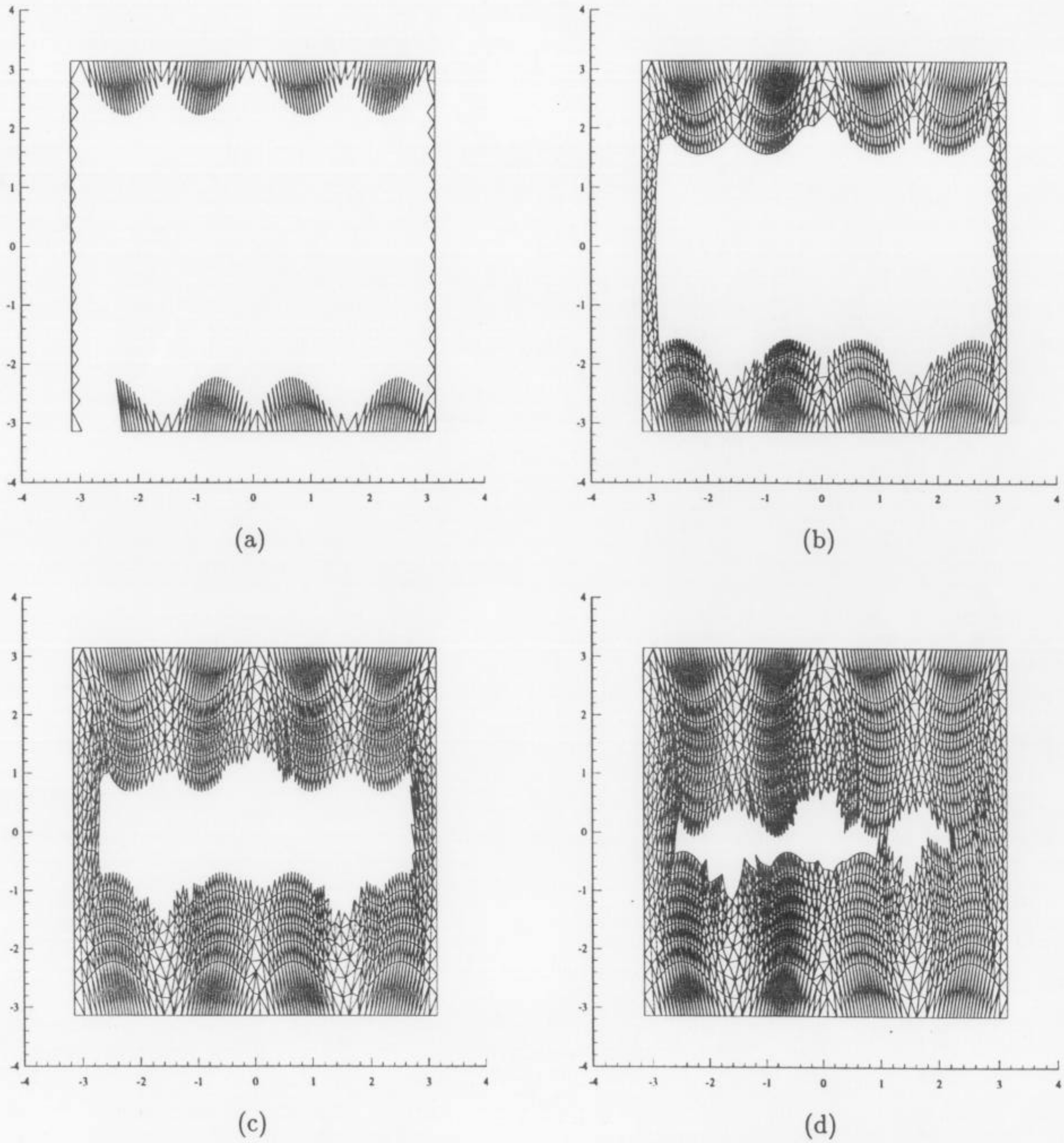


Figure 5: Advancing front in parameter space Σ with 185, 1000, 2000, and 2650 triangles. Note how in (d) the front has pinched itself closed, generating two child fronts.

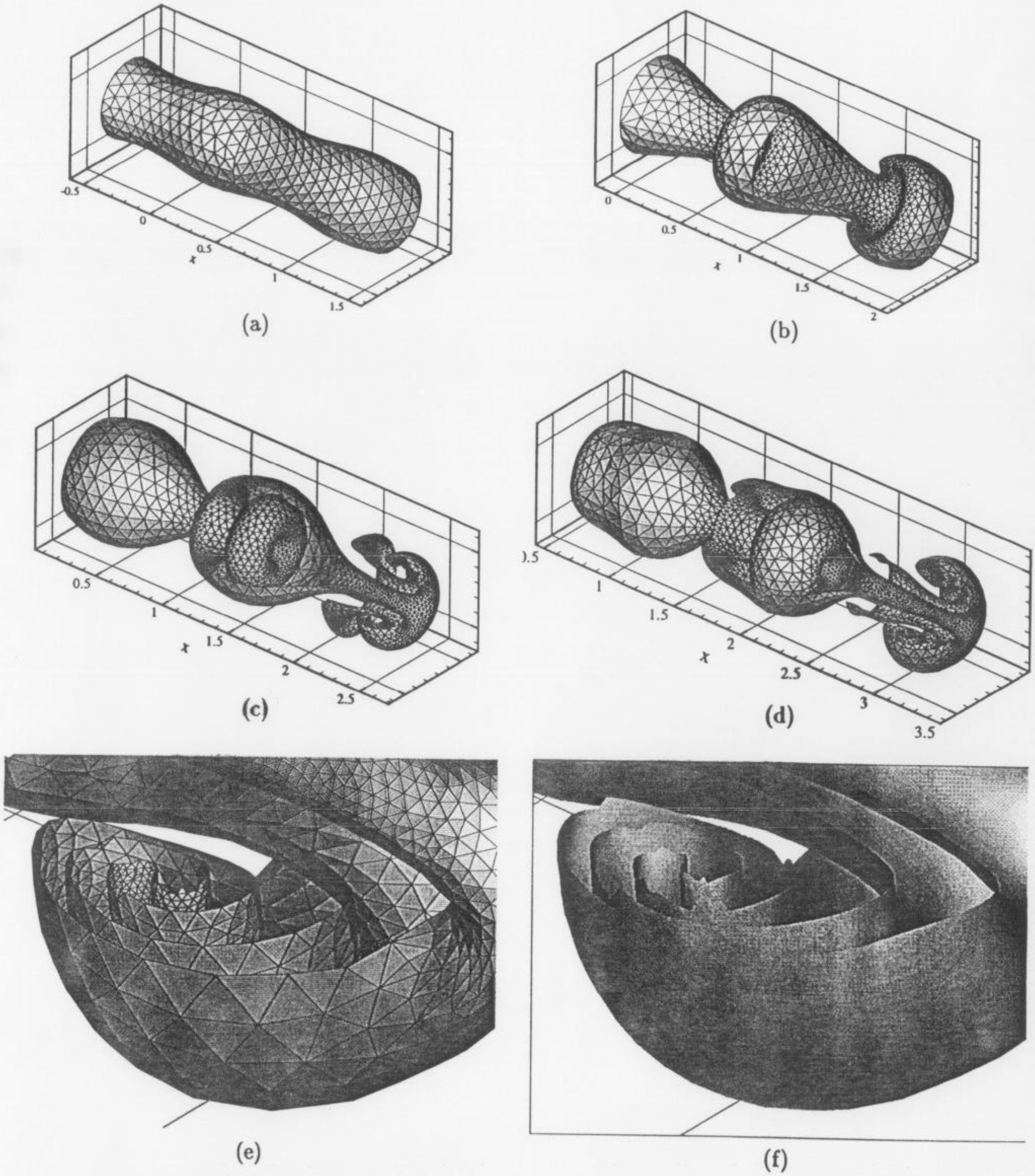


Figure 11: Periodic-train vortex sheet with strength distribution simulating a hollow-core jet with initial 1^{st} mode axial perturbation, $\sigma = 0.2$ (relative to initial average diameter). One period calculated, two shown with second in cut-away. Mesh keleton: (a) t=0; (b) t=0.2; (c) t=0.4; (d) t=0.6; (e) zoom of roll-up region in (d); (f) actual smooth surface representation of (e).

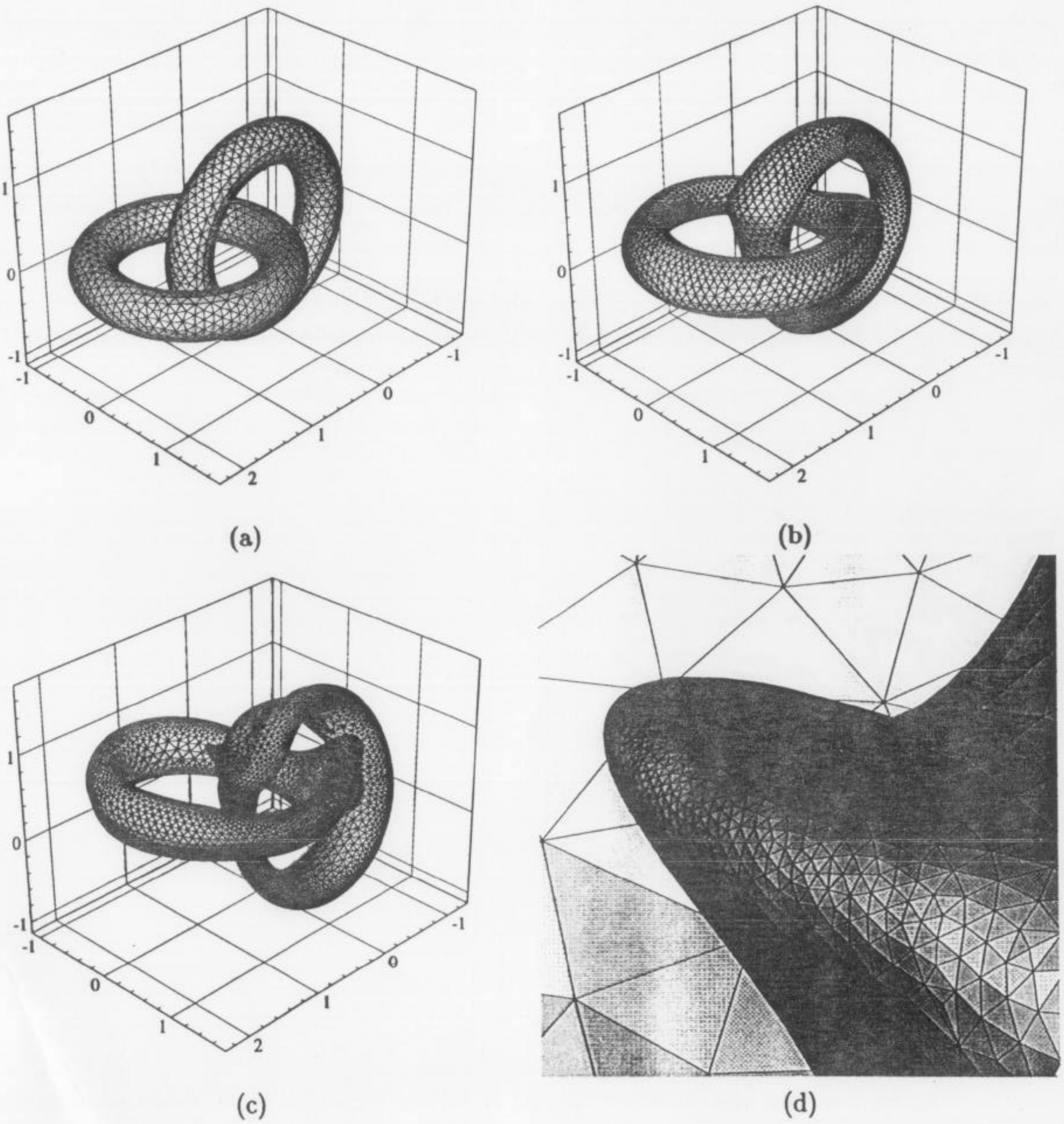


Figure 13: Mesh skeleton of interlocked hollow-core vortex rings with perpendicular impulse vectors, $\sigma = 0.2$. Ring to tube radius ratio is 4. Vertical ring impulse points right and out of page, while orizontal ring impulse points up. (a) t=0, (b) t=0.25, (c) t=0.5, (d) zoom of center region in (c).